Wind Induced Vibrations of Pole Structures

A Project Report

Presented to

the Department of Civil and Geological Engineering

Faculty of Engineering

The University of Manitoba

In Partial Fulfillment

of the Requirements for the Degree

Master of Science in Civil Engineering

by

Wayne Flather

June 1997

0-612-23302-2

THE UNIVERSITY OF MANITOBA

FACULTY OF GRADUATE STUDIES

COPYRIGHT PERMISSION

WIND INDUCED VIBRATIONS OF POLE STRUCTURES

by

WAYNE FLATHER

A Thesis submitted to the Faculty of Graduate Studies of the University of Manitoba in partial fulfillment of the requirements of the degree of

MASTER OF SCIENCE

ABSTRACT

A user friendly, interactive computer program was created to provide a better understanding of street light structures used by Manitoba Hydro. The need for such a program arose after several failures of such structures. In order to understand these failures, an understanding of two wind conditions which possibly caused the failures was required. The two wind conditions of interest are low speed laminar winds, causing vortex shedding, and gust winds causing vibrations parallel to the direction of the wind. Forcing functions were developed, based on common fluid dynamic theories, which are used to model the forces exerted on the pole due to these wind conditions. A mathematical model was built in order to determine the response of a pole subjected to these types of wind conditions. The model is analyzed using the finite element method and common mathematical routines. The computer program allows the user to vary parameters relating to both the pole structure and the wind conditions. By varying the parameters and observations of the graphical display of the expected pole vibrations, an in depth understanding of the pole's structural behavior can be achieved. This study can be applied to the future design of pole structures as well as to continued maintenance and monitoring programs.

ACKNOWLEDGMENTS

I am deeply grateful to my advisor, Professor A.H. Shah, for his academic support and guidance as well as experience and knowledge throughout this work. I am also grateful to Manitoba Hydro Engineers, Mr. D. Spangelo and Mr. G. Penner, for their suggestion of the research topic, helpful discussions and expertise in this work.

I would also like to thank Professor N. Popplewell and Dr. J. Frye for serving as examiners.

The helpful advise in writing and editing the report provided by Ms. C. Lodge and the helpful computer advise provided by Mr. J. Rogers is greatly appreciated.

The financial support provided by Manitoba Hydro is gratefully acknowledged.

Finally, a special thanks to my parents, my family, my friends and especially Colleen for their general support and patience throughout the course of my M.Sc. program.

Contents

List of Tables

List of Figures

Chapter 1

Introduction

1.1 Purpose

The purpose of this study is to create a user friendly, interactive computer program for Manitoba Hydro. This program will provide a better understanding of the effects wind has on street light structures currently in use by Manitoba Hydro. The specific wind conditions under investigation are low speed laminar winds, that cause vortex shedding, and gust winds. Investigation of these wind conditions and the resulting response of the structure, provides insight into the potential failure mechanisms. A better understanding of the behavior of these street light structures due to these wind conditions may aid in better future design practices and an awareness of areas of potential concern for monitoring the structures.

The program allows the user to vary any number of variables pertaining to the structure and wind conditions including the structure's geometry, material properties as well as the wind direction, laminar wind speed and gust wind data.

1.2 Scope

The inspiration for this study arose when several pole structures failed due to fatigue believed to be caused by vibrations induced by wind loading. See Figures 1.1 and 1.2. In order to better understand any phenomenon involving the interaction of two systems, in this case the pole structures and wind forces, it is beneficial to have a graphical representation of the problem. An interactive computer program in which the user could observe the predicted wind induced vibrations of the structure while altering the variables was deemed to be the best solution.

Figure 1.1: Photograph of failed street light pole.

The scope of this report includes a description of the theory used to develop the loading due to the two wind conditions. This is followed by an explanation of the method used to determine the response due to these loads and a numerical example used to demonstrate these theories.

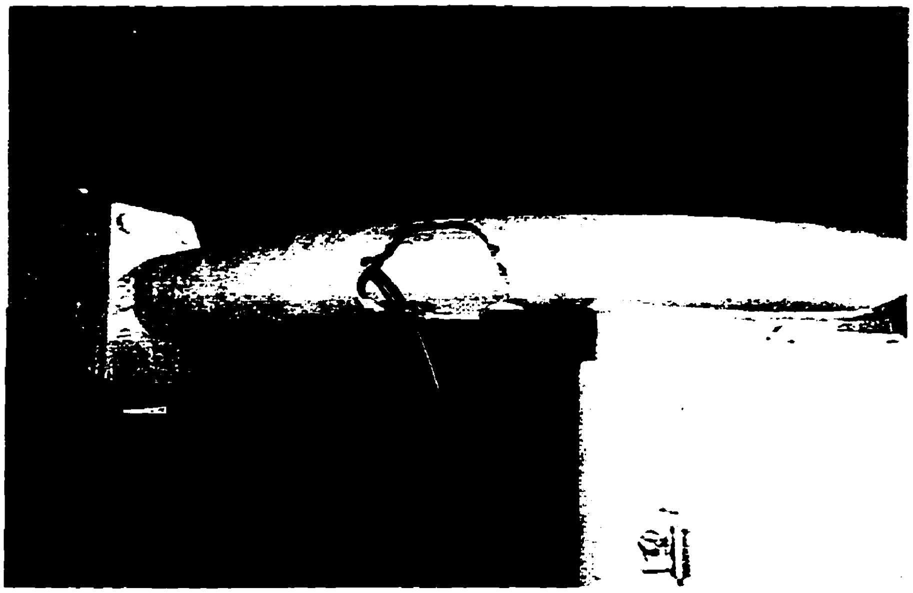

Figure 1.2: Photograph of failed luminaire assembly.

1.3 Overview of the Present Study

The first part of this report describes the theory of wind engineering, including a description of commonly used constants used in fluid mechanics for bluff bodies. This section is included to develop the forcing functions which are used to model the forces exerted on the light pole due to the two wind conditions.

Much research focusing on wind induced vibrations of poles and towers has been done over the past two decades, [1]-[6]. Most notably is the work done by Mehta [1], in which full scale field tests and tow tank tests were done to gain a better understanding of the interaction between wind and pole structures. Many books have also been written on the topic of wind engineering and fluid dynamics, [7]-[10], which explain the theories used by this study.

Following the review of the fundamentals of wind engineering is a description of the process, theory and rationale used to build the mathematical model of the structure. The finite element method, which was used to build the mathematical model, is explained, including a description of the elements used. Many books have been written on the topic of the finite element method, [11]-[13], which explain the theories used by this study.

The next section describes the method by which the response of the mathematical model due to the forcing functions is determined. This is done by determining the free response of the structure resulting in natural frequencies and associated mode shapes. Forcing functions are then applied to the model and the response is determined by employing commonly used mathematical routines. Many books have been written on the topic of structural dynamics, [14]-[17], which explain the theories used by this study.

Finally, a description of a numerical example of a typical street light pole, [18], is presented. This section presents the mathematical model, the typical wind loading and the response due to that loading. Also, this section provides a demonstration of the capabilities of the program which was written.

Included at the end of the report are a number of appendices. These appendices include a manual for the computer program, examples used to verify the program, a listing of the FORTRAN code and a list of the nomenclature used in the report.

Chapter 2

Wind Loading on Structures

In the past four decades there have been many interesting advances in the application of aerodynamics in the field of civil engineering. These applications are limited mainly to relatively low-speed, incompressible flow phenomena, such as aeolian vibration and galloping. Winds acting on a cylindrically shaped structure, for example, may induce forces in several different directions. Wind forces naturally cause a deflection in the direction parallel to the wind direction. Vibrations perpendicular to the wind direction can also be induced in certain instances. Perpendicular vibration is induced by the phenomenon known as aeolian vibration, more commonly called vortex shedding. Vortex shedding occurs in laminar winds at relatively low wind speeds, when the air stream separates on each side of a structure and vortices or eddies are formed alternately at each separation edge. The formation and detachment of each eddy induces a suction force at the separation points. The suction force alternates back and forth between the points of formation of the eddies.

Another common phenomenon discussed in wind engineering is known as galloping. Galloping causes vibrations of very large amplitudes at low frequencies. Galloping is an instability of slender structures with certain asymmetrical cross sections such as rectangular or D sections. The frequencies of the perpendicular oscillations caused by galloping are much lower than those caused by vortex shedding.

This report focuses on gust winds causing oscillations parallel to a wind and laminar winds causing oscillations perpendicular to the wind. The galloping phenomenon is not discussed further. This chapter examines the fundamental theory associated with gust winds and laminar winds causing vortex shedding and the forces which they exert on a structure.

2.1 Reynolds Number

Air flow around a structure is dependent on many variables such as flow velocity, cross sectional area, surface roughness and wind direction. The Reynolds number is a non-dimensional index that helps to predict expected flow characteristics. The Reynolds number, Re, is defined as,

$$Re = \frac{Vd}{\nu}, \tag{2.1}$$

where V is the free stream wind speed, d is the projected width of the structure perpendicular to the wind flow commonly referred to as the bluff diameter and ν is the kinematic viscosity of air.

2.2 Wake and Vortex Formations

Air flow around a structure is dependent on many variables; one of which is the cross section of the structure. Pole structures are assumed here to have a circular cylinder shape. Therefore, a brief discussion of the air flow characteristics around a circular cylinder is needed.

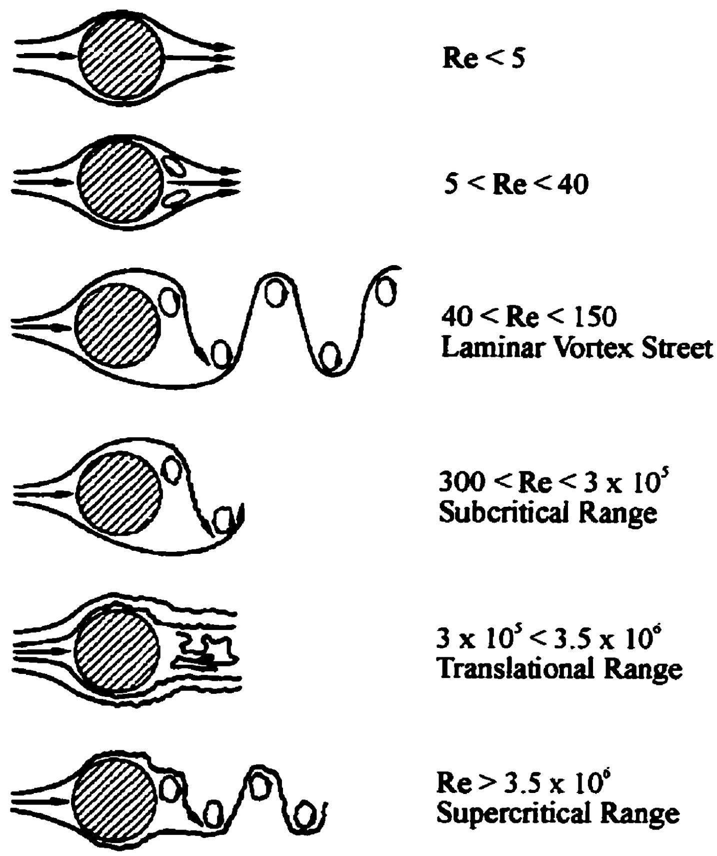

Figure 2.1: Flow past a circular cylinder, [7].

The major Reynolds number ranges for a smooth circular cylinder are given in Figure 2.1. At very low Reynolds numbers, $Re < 5$, the fluid flow follows the cylinder contours. In the range $5 < Re < 40$, the flow separates from the back of the cylinder and a symmetrical pair of vortices is formed in the wake. The length of the vortices increases linearly with Reynolds number, reaching a length of three cylinder diameters at a Reynolds number of 40. As the Reynolds number is increased further, the wake becomes unstable and one of the vortices breaks away. A laminar periodic wake of staggered vortices is formed downstream of the cylinder which is called the vortex street. Between the range $Re = 150$ and 300, the vortices breaking away from the cylinder become turbulent.

The Reynolds number range $300 < Re < 3 \times 10^5$ is called subcritical. In this range, the vortex shedding is strong and periodic. In the transitional range, $3 \times 10^5 < Re < 3.5 \times 10^6$, the wake is narrow and disorganized. In the supercritical Reynolds number range, $Re > 3.5 \times 10^6$, regular vortex shedding is re-established. However, the wake is now turbulent.

This study deals with vortex shedding in the subcritical range. For pole diameters between 100 and 500 mm, this relates to wind speeds below 10 m/s.

2.3 Strouhal Number

The Strouhal number is a nondimensional number which defines the regularity of the vortex wake effects described previously. The Strouhal number, S, is defined as,

$$S = \frac{f_s d}{V}, \tag{2.2}$$

where f_s is the vortex shedding frequency in cycles/s (Hz).

The Strouhal number of a stationary circular cylinder is a function of Reynolds number and surface roughness, as shown in Figure 2.2. The Strouhal number follows the Reynolds number flow ranges of Figure 2.1. In the Reynolds number range of $3 \times 10^5 < Re < 3.5 \times 10^6$, very smooth surface cylinders have a chaotic, disorganized, high frequency wake and Strouhal numbers are as high as 0.5. Rough cylinders have organized periodic wakes with Strouhal numbers of approximately 0.25. For Reynolds numbers below this range, vortex induced vibration of cylinders generally occurs at Strouhal numbers of approximately 0.2.

As stated previously, this study focuses on the subcritical Reynolds number range which corresponds to a Strouhal number of about 0.2.

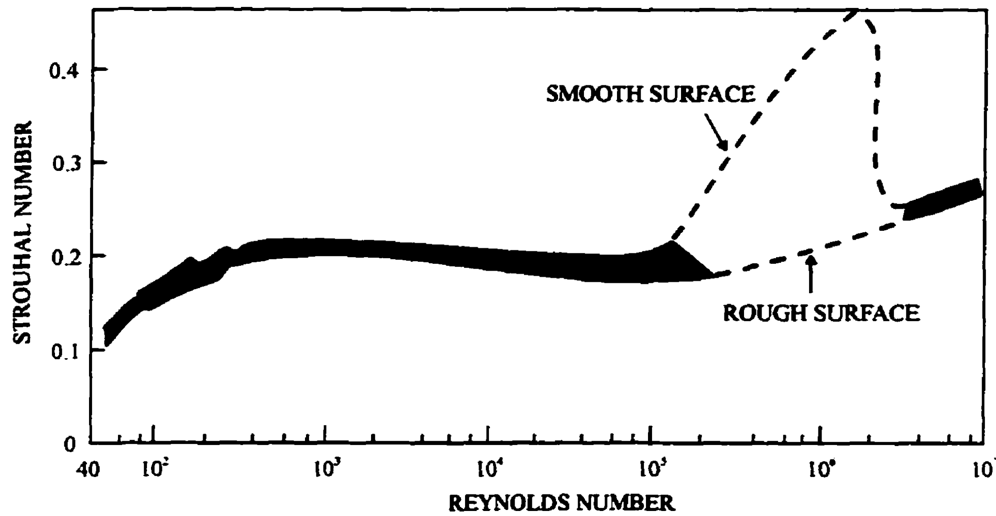

Figure 2.2: Strouhal number and Reynolds number relationship for circular cylinders, [7].

2.4 Wind Forces

As stated previously, this report focuses on along wind forces, due to gust winds and across wind forces, due to vortex shedding. These forces can be defined by the use of the same general forcing function,

$$F = \frac{1}{2}\rho_{air} V^2 A, \tag{2.3}$$

where F is the wind force, ρ_{air} is the mass density of air and A is the area upon which the wind force acts.

To model the oscillating, perpendicular forces created by vortex shedding, Equation 2.3 can be modified by multiplying by a sine term to obtain,

$$F_L = \frac{1}{2}\rho_{air} V^2 A \sin(\omega_s t), \tag{2.4}$$

where $\omega_s = 2\pi f_s$ is the circular vortex shedding frequency (rads/s) and t is time (s).

The along wind forces due to gust loads is calculated as follows,

$$F_D = \frac{1}{2}\rho_{air} (V(t))^2 A. \tag{2.5}$$

It should be noted that the wind speed term in Equation 2.4 is not a function of time and therefore, Equation 2.4 will only produce forces if the wind speed is held constant. This is in agreement with past studies [8] which have shown that vortex oscillations are usually excited only when the wind speed is unidirectional and constant.

This study focuses on the aerodynamic forces due to wind flowing around the pole structure only. Additional aerodynamic forces resulting from wind flowing around elements other than the pole, such as luminaires, are neglected.

Figure 2.3 shows a bluff body and the resulting along wind force, F_D, and across wind force, F_L.

2.5 Inclined Structures

Up to this point of the discussion it has been assumed that the flow direction is at right angles to the structure. Inclined cylinders affect both Equations 2.4 and 2.5.

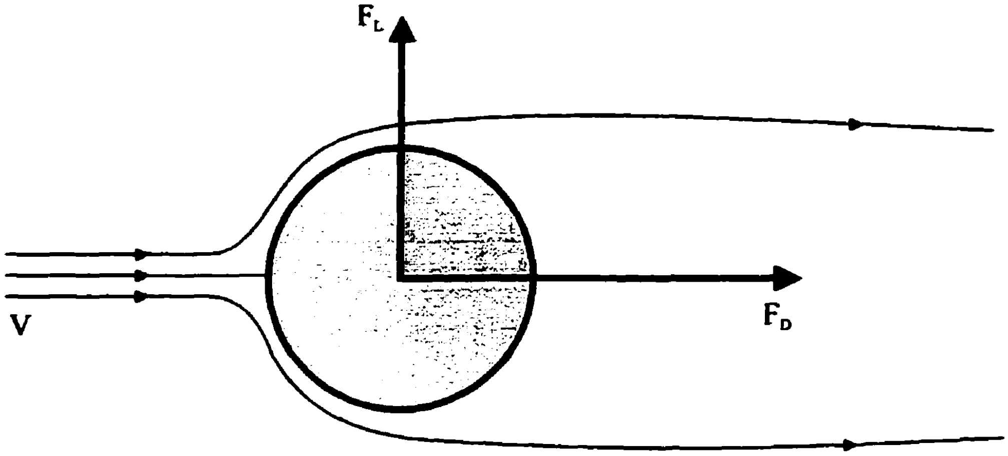

Figure 2.3: Lift and drag forces, F_L and F_D respectively, acting on a bluff body.

When determining, A, the area upon which the wind force acts, the projected area of the cylinder perpendicular to the wind flow must be used. This is illustrated in Figure 2.4. If L is the length of the element and θ is the angle of inclination to the wind flow, the resulting projected area is,

$$A = dL\sin\theta. \tag{2.6}$$

A structure's inclination to the wind flow also has an effect on the shedding frequency, f_s in Equation 2.2. The shedding frequency is determined by using the perpendicular wind speed component, $V\sin\theta$, as shown in Figure 2.4. The resulting shedding frequency is,

$$f_s = \frac{SV\sin\theta}{d}. \tag{2.7}$$

2.6 Tapered Structures

In Equations 2.2 and 2.7 the d term refers to the bluff diameter of the cross section. Most pole structures are tapered, having a larger diameter at the base than at the

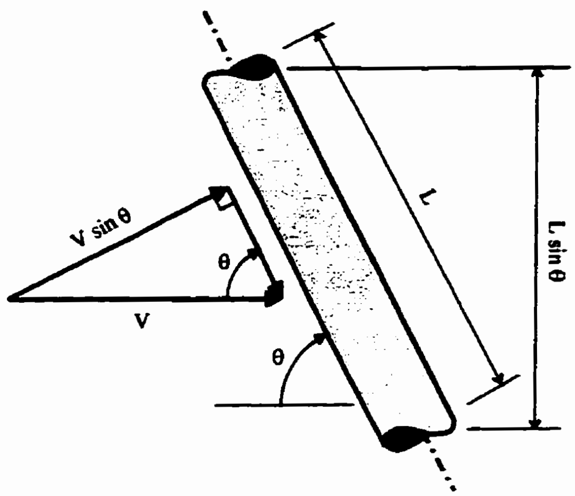

Figure 2.4: Structure inclined to wind.

tip. This varying diameter results in the shedding frequency, f_s, varying over the length of the pole. This is undesirable in the analysis procedure, because the lift force, Equation 2.4, will also vary along the length of the pole. There has been some question as to what value of d should be used . It has been recommended [2] that the diameter at the tip of tapered poles should be used. This, in effect, lowers the shedding frequency.

Chapter 3

Structural Model

The formation of a mathematical model of the structure is an important stage of any engineering analysis. The model allows the displacement of the structure to be described in terms of the stiffness and mass of the structure and the forces applied to the structure. For the present study, the pole structure is modeled mathematically by using the finite element method (FEM). The finite element method, is a computer aided mathematical technique used to obtain an approximate numerical solution to equations that predict the response of a physical system under external influences.

The first step in creating a finite element model is to discretize the structure by subdividing the body into an equivalent system of small bodies called finite elements. This procedure is illustrated in Figure 3.1. The points at which the primary unknowns are required to be evaluated, are called nodes or nodal points. The number of unknowns at a node is the nodal degrees-of-freedom (DOF).

The selection of the element type and number of elements to be used has a significant impact on the accuracy of the solution as well as the computational effort. In this study, the three dimensional pole structure is modeled by two separate two dimensional models. This is made possible by assuming that the pole exists in a single plane, defined by two domains, X and Y, see Figure 3.1. The *in-plane* displacement is modeled by a prismatic plane frame element, which has three DOF at each node. The nodal DOF for the plane frame element are, translation in the X and Y direction and rotation about the Z direction (normal to the $X-Y$ plane). The *out-of-plane* displacement is modeled by a prismatic grid element, which also has three DOF at each node. The nodal DOF for the grid element are translation in the Z direction and rotation about the X and Y direction.

This chapter further explains the two elements used to build the finite element model.

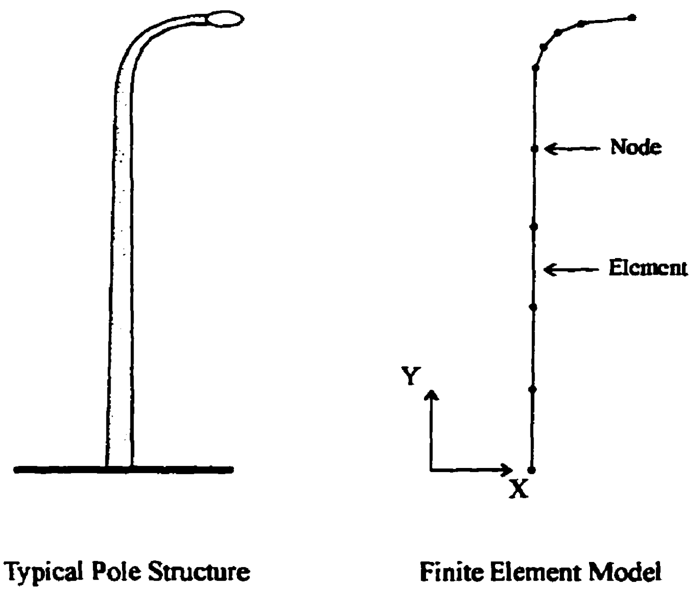

Figure 3.1: Discretized finite element model of a typical lighting pole.

3.1 The Plane Frame Element

A plane frame element is a series of slender elements connected rigidly to each other; that is, the original angles between elements at their joints remain unchanged after deformation. Furthermore, moments are transmitted from one element to another at the joints. Hence, moment continuity exists at the rigid joints. In addition, the element centroids, as well as the applied loads, lie in a common plane.

Figure 3.2 shows an arbitrarily oriented plane frame element. The x and y axes are the local coordinate system, having the origin located at node 1 and the x axes located along the length of the element (node $1 \rightarrow 2$). The global axes X and Y are chosen with respect to the whole structure. The local nodal DOF, u, v and ϕ_z, correspond, respectively to the axial, shear and flexural deformations.

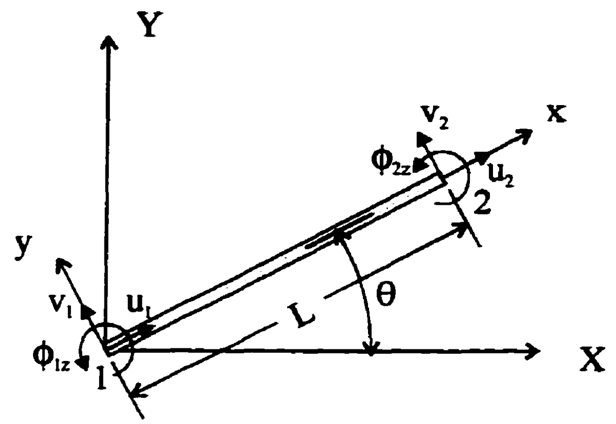

Figure 3.2: Arbitrarily oriented plane frame element.

The elemental stiffness matrix in local co-ordinates, $[k_e]$, for a plane frame element is, [12],

$$[k_e] = \begin{bmatrix} C_1 & 0 & 0 & -C_1 & 0 & 0 \\ 0 & 12C_2 & 6C_2L & 0 & -12C_2 & 6C_2L \\ 0 & 6C_2L & 4C_2L^2 & 0 & -6C_2L & 2C_2L^2 \\ -C_1 & 0 & 0 & C_1 & 0 & 0 \\ 0 & -12C_2 & -6C_2L & 0 & 12C_2 & -6C_2L \\ 0 & 6C_2L & 2C_2L^2 & 0 & -6C_2L & 4C_2L^2 \end{bmatrix}, \quad (3.1)$$

where $C_1 = \frac{AE}{L}$, A is the cross sectional area of the element, E is the modulus of elasticity of the material, L is the length of the element, $C_2 = \frac{EI}{L^3}$ and I is the moment of inertia of the element.

The local displacements are related the global displacements, d_X, d_Y and ϕ_Z, by a transformation matrix, $[T]$, given by,

$$[T] = \begin{bmatrix} C & S & 0 & 0 & 0 & 0 \\ -S & C & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & C & S & 0 \\ 0 & 0 & 0 & -S & C & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix} \tag{3.2}$$

where $C = \cos\theta$ and $S = \sin\theta$.

The elemental stiffness matrix in global co-ordinates, $[K_e]$, can be determined by substituting Equations 3.1 and 3.2 into the following Equation,

$$[K_e] = [T]^T [k_e] [T] . \tag{3.3}$$

The elemental mass matrix in local coordinates, $[m_e]$, for a plane frame element is, [12],

$$[m_e] = \frac{\rho AL}{420} \begin{bmatrix} 140 & 0 & 0 & 70 & 0 & 0 \\ 0 & 156 & 22L & 0 & 54 & -13L \\ 0 & 22L & 4L^2 & 0 & 13L & -3L^2 \\ 70 & 0 & 0 & 140 & 0 & 0 \\ 0 & 54 & 13L & 0 & 156 & -22L \\ 0 & -13L & -3L^2 & 0 & -22L & 4L^2 \end{bmatrix} . \tag{3.4}$$

where ρ is the mass density of the material.

Similarly, the elemental mass matrix in global coordinates, $[M_e]$, is determined by substituting Equation 3.4 and 3.2 into the following Equation,

$$[M_e] = [T]^T [m] [T] . \tag{3.5}$$

3.2 The Grid Element

A grid is a structure on which loads are applied perpendicular to the plane of the structure, as opposed to a plane frame where the loads are applied in the plane of the structure. The elements of a grid are assumed to be connected rigidly so that the original angles between elements connected together at a node remain unchanged. Both torsional and bending moment continuity then exist at the node point of a grid.

Figure 3.3 shows an arbitrarily oriented grid element. The local and global co-ordinate systems, ($x - y$ and $X - Y$), are the same as for the plane frame element. However, an additional domain must be defined for the *out-of-plane* direction, which is noted as z in the local co-ordinate system and Z in the global system. The local nodal DOF, w, ϕ_x and ϕ_y correspond, respectively, to shear, torsional and flexural deformations.

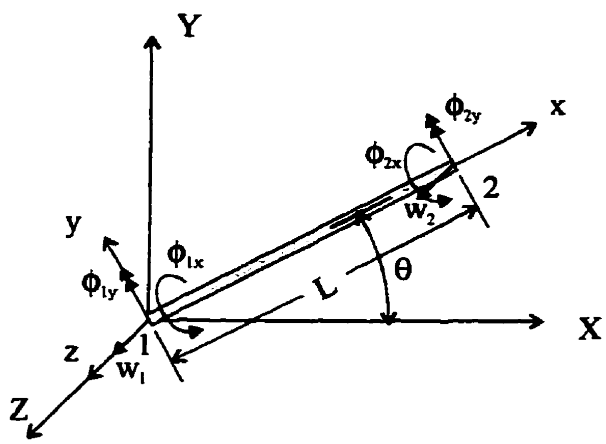

Figure 3.3: Arbitrarily oriented grid element.

The elemental stiffness matrix in local co-ordinates, $[k_e]$, for a grid element is, [12],

$$[k] = \begin{bmatrix} \frac{12EI}{L^3} & 0 & \frac{6EI}{L^2} & \frac{-12EI}{L^3} & 0 & \frac{6EI}{L^2} \\ 0 & \frac{GJ}{L} & 0 & 0 & -\frac{GJ}{L} & 0 \\ \frac{6EI}{L^2} & 0 & \frac{4EI}{L} & \frac{-6EI}{L^2} & 0 & \frac{2EI}{L} \\ \frac{-12EI}{L^3} & 0 & \frac{-6EI}{L^2} & \frac{12EI}{L^3} & 0 & \frac{-6EI}{L^2} \\ 0 & -\frac{GJ}{L} & 0 & 0 & \frac{GJ}{L} & 0 \\ \frac{6EI}{L^2} & 0 & \frac{2EI}{L} & \frac{-6EI}{L^2} & 0 & \frac{4EI}{L} \end{bmatrix}; \qquad (3.6)$$

where G is the shear modulus and J is the polar moment of inertia of the element.

The elemental mass matrix in local co-ordinates, $[m_e]$, for a grid element is, [12],

$$[m_e] = \frac{\rho AL}{420} \begin{bmatrix} 156 & 0 & 22L & 54 & 0 & -13L \\ 0 & 140 & 0 & 0 & 70 & 0 \\ 22L & 0 & 4L & 13L & 0 & -3L^2 \\ 54 & 0 & 13L & 156 & 0 & -22L \\ 0 & 70 & 0 & 0 & 140 & 0 \\ -13L & 0 & -3L^2 & -22L & 0 & 4L^2 \end{bmatrix} \qquad (3.7)$$

The global stiffness and mass matrices are determined by substituting Equations 3.6 and 3.7 into Equations 3.3 and 3.5, respectively. However, the transformation matrix, $[T]$, for a grid element is,

$$[T] = \begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 \\ 0 & C & S & 0 & 0 & 0 \\ 0 & -S & C & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 & C & S \\ 0 & 0 & 0 & 0 & -S & C \end{bmatrix}. \qquad (3.8)$$

3.3 Additional Masses

The previously mentioned mass matrices account for the mass of the structural (load bearing) components only. However, it may also be desired to model the mass of non-structural components as well, such as luminaries or traffic signs. This is accomplished by adding a lumped (point) mass at the appropriate node. If a point mass is added to the plane frame element shown in Figure 3.2, the resulting elemental mass matrix is,

$$[m_e] = \frac{\rho AL}{420}\begin{bmatrix} 140 & 0 & 0 & 70 & 0 & 0 \\ 0 & 156 & 22L & 0 & 54 & -13L \\ 0 & 22L & 4L^2 & 0 & 13L & -3L^2 \\ 70 & 0 & 0 & 140 & 0 & 0 \\ 0 & 54 & 13L & 0 & 156 & -22L \\ 0 & -13L & -3L^2 & 0 & -22L & 4L^2 \end{bmatrix} + \begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & M_L & 0 & 0 \\ 0 & 0 & 0 & 0 & M_L & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix}. \tag{3.9}$$

where M_L is the total mass of the non-structural component. Notice that an additional mass is added only to the translational DOF for node 2, the effect on the rotational DOF is assumed to be negligible. Similarly, for the grid element additional masses are added to the translational DOF only.

Chapter 4

Dynamic Analysis

This chapter discusses the procedure by which the dynamic response of the finite element model is determined.

4.1 Free Vibration Analysis

When a system is displaced from its static equilibrium position and then released, it vibrates freely about its equilibrium position with a behavior that depends upon the mass and stiffness of the system. The purpose of a free vibration analysis is to determine this behavior in terms of the frequencies, known as the natural frequencies, ω_n, and the associated deformed shapes, known as mode shapes, ϕ.

4.1.1 Procedure

As discussed previously, a structure can be divided into discrete elements and the equations of motion can be written for each DOF. These equations can be written in matrix form as, [16],

$$[M]\left\{\ddot{X}\right\}+[C]\left\{\dot{X}\right\}+[K]\{X\}=\{F(t)\} \tag{4.1}$$

where $[M]$, $[C]$, $[K]$ are the global mass, damping and stiffness matrices, respectively, $\left\{\ddot{X}\right\}$,$\left\{\dot{X}\right\}$,$\{X\}$ are the acceleration, velocity and displacement vectors, respectively, and $\{F(t)\}$ is the forcing vector.

For undamped free vibrations, the forcing vector is set to the null vector and the damping matrix is neglected. Equation 4.1 reduces to,

$$[M]\left\{\ddot{X}\right\}+[K]\{X\}=\{0\}\,. \tag{4.2}$$

If a simple harmonic motion is assumed for each DOF and substituted into Equation 4.2, the resulting equation can be manipulated to the following form,

$$[M]^{-1}[K]\{X\}=\omega^2\{X\}\,. \tag{4.3}$$

Equation 4.3 is now in the form of a real, general eigenvalue problem. The solution yields the eigenvalues which are equal to the natural frequencies squared, ω^2, and the eigenvectors which are equal to the mode shapes, ϕ.

4.2 Forced Vibration Analysis

Forced vibration occurs when a system is subjected to an external excitation that adds energy to the system, such as a wind exerting excitation forces on a cylinder. (See Chapter 2). In general, the amplitude of such a vibration depends upon the natural frequencies of the system and the damping inherent in the system, as well as upon the frequency components present in the exciting force. The amplitude of a forced vibration can become very large when a frequency component of the excitation approaches one of the natural frequencies of the system. Such a condition is referred to as resonance, and the resulting stresses and strains have the potential of causing failures.

4.2.1 Modal Analysis

Modal analysis involves the decoupling of the differential equations of motion as a means of reducing a multiple DOF system to a number of independent, single DOF systems. The single DOF systems that result from the decoupling process are expressed in terms of principal coordinates. The principal coordinates are independent of the original system. The response of each single DOF system can be determined and then superimposed to find the response of the original system.

The general form of the matrix equation for an n DOF system can be found from Equation 4.1. The first step in decoupling this matrix equation is to transform the displacement, velocity and acceleration vectors to the principal coordinates. This is done by using the following equations;

$$\{X\} = [\Phi]\{\delta\}, \tag{4.4}$$

$$\left\{\dot{X}\right\} = [\Phi]\left\{\dot{\delta}\right\} \tag{4.5}$$

and

$$\left\{\ddot{X}\right\} = [\Phi]\left\{\ddot{\delta}\right\}, \tag{4.6}$$

where $[\Phi]$ is the modal matrix, in which each column represents a mode shape, ϕ, and $\{\delta\}$, $\left\{\dot{\delta}\right\}$ and $\left\{\ddot{\delta}\right\}$ are the displacement, velocity and acceleration vectors, respectively, in terms of the principal coordinates. Substituting Equations 4.4, 4.5 and 4.6 into Equation 4.1 results in,

$$[M][\Phi]\left\{\ddot{\delta}\right\} + [C][\Phi]\left\{\dot{\delta}\right\} + [K][\Phi]\{\delta\} = \{F(t)\}. \tag{4.7}$$

Multiplying this equation by the transpose of the modal matrix, $[\Phi]^T$ produces,

$$[\mathbf{M}]\left\{\ddot{\delta}\right\}+[\mathbf{C}]\left\{\dot{\delta}\right\}+[\mathbf{K}]\{\delta\}=\{\mathbf{F}(t)\}, \tag{4.8}$$

where $[\mathbf{M}] = [\Phi]^T[M][\Phi]$, $[\mathbf{K}] = [\Phi]^T[K][\Phi]$, are the diagonal modal mass and stiffness matrices, respectively, $[\mathbf{C}] = [\Phi]^T[C][\Phi]$ is the general modal damping matrix and $\{\mathbf{F}(t)\} = [\Phi]^T\{F(t)\}$ is the modal force vector. This equation can be simplified further by writing the modal damping matrix in terms of the modal mass matrix and assuming Rayleigh Damping [16]. For the r^{th} mode the damping constant can be written as,

$$C_r = 2\zeta_r\omega_r M_r, \tag{4.9}$$

where C_r and M_r are the diagonal component of the modal damping and mass matrix, respectively, ζ_r is the modal damping factor and ω_r is the natural frequency for the r^{th} mode. Also, the modal stiffness matrix can be written in terms of the modal mass matrix by,

$$K_r = \omega_r^2 M_r. \tag{4.10}$$

The resulting equation of motion for the r^{th} mode can be written as,

$$\ddot{\delta}_r + 2\zeta_r\omega_r\dot{\delta}_r + \omega_r^2\delta_r = \frac{F_r(t)}{M_r} = E_r(t), \tag{4.11}$$

where $F_r(t)$ is the modal forcing function and $E_r(t)$ is the excitation function of the r^{th} mode.

4.2.2 Response of a System Subjected to a General External Force

Determining the response of a system subjected to a general, time dependent force involves finding the solution to Equation 4.11. The numerical method used in this study for integrating the differential equations is the widely used fourth-order Runge-Kutta method.

The first step for numerically integrating Equation 4.11, is to rewrite the equation in terms of its highest-order derivative as $\ddot{\delta} = f\left(t, \delta, \dot{\delta}\right)$, which results in,

$$\ddot{\delta} = -2\zeta\omega\dot{\delta} - \omega^2\delta + E\left(t\right). \tag{4.12}$$

Notice that the subscript r has been dropped in the notation, in order to simplify the notation. The Runge-Kutta recurrence formulae for solving Equation 4.12 in terms of the step size Δt are,

$$\delta_{i+1} = \delta_i + \Delta t\dot{\delta}_i + \frac{\Delta t}{6}\left(k_1 + k_2 + k_3\right) + 0\left(\Delta t^2\right) \tag{4.13}$$

and

$$\dot{\delta}_{i+1} = \dot{\delta}_i + \frac{1}{6}\left(k_1 + 2k_2 + 2k_3 + k_4\right) + 0\left(\Delta t^2\right) \tag{4.14}$$

where

$$\begin{aligned} k_1 &= \Delta t\ f\left(t_i, \delta_i, \dot{\delta}_i\right) \\ k_2 &= \Delta t\ f\left(t_i + \tfrac{\Delta t}{2}, \delta_i + \tfrac{\Delta t}{2}\dot{\delta}_i, \dot{\delta}_i + \tfrac{k_1}{2}\right) \\ k_3 &= \Delta t\ f\left(t_i + \tfrac{\Delta t}{2}, \delta_i + \tfrac{\Delta t}{2}\dot{\delta}_i + \tfrac{\Delta t}{4}k_1, \dot{\delta}_i + \tfrac{k_2}{2}\right) \\ k_4 &= \Delta t\ f\left(t_i + \Delta t, \delta_i + \Delta t\dot{\delta}_i + \tfrac{\Delta t}{2}k_2, \dot{\delta}_i + k_3\right). \end{aligned} \tag{4.15}$$

The numerical solution begins with the substitution of the initial values of δ and $\dot{\delta}$ into Equation 4.12 to obtain a value of the function $f\left(t, \delta, \dot{\delta}\right)$ for use in determining k_1. The values for k_2, k_3 and k_4 are then determined successively for use in the recurrence formulae of Equations 4.13 and 4.14 to obtain values of δ_{i+1} and $\dot{\delta}_{i+1}$. The latter are then used in Equations 4.12 and 4.15 to obtain new k_1, k_2, k_3 and k_4 values for substitution into Equations 4.13 and 4.14 to obtain δ_{i+2} and $\dot{\delta}_{i+2}$, and so on.

The solution from the numerical integration determines the time response of the displacement, velocity and acceleration in terms of the principal coordinates. Substituting these solutions into Equations 4.4, 4.5 and 4.6 yields the displacement, velocity and acceleration of the original system, respectively.

4.2.3 Response of a System Subjected to a Harmonic External Force

When a system is disturbed by a temporally periodic external force, the resulting response of the system can be considered to be the sum of two distinct components, the forced response and the free response. The forced response resembles the exciting force in its mathematical form. The free response does not depend on the characteristics of the exciting function but only upon the physical parameters of the system itself.

In any system disturbed by a sinusoidal excitation, or any temporally periodic excitation, the free response that is initiated when the excitation is first applied dies out with time because of the inherent damping in the system. Eventually only the forced response remains. Because the free response of the system dies out with time, it is often referred to as a transient response. The forced response is known as the steady-state response. This is shown clearly in Figure 4.1 which presents part of a plot of the response of a system due to a sinusoidal excitation.

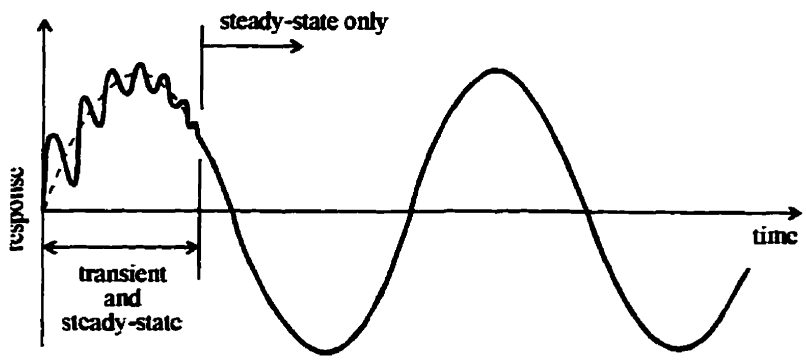

Figure 4.1: Response of a system due to sinusoidal excitation.

Consider a system which is excited by a sinusoidal excitation such as,

$$F(t) = B \sin(\Omega t), \tag{4.16}$$

where B is the amplitude of the excitation and Ω is the frequency of the excitation. Using modal analysis, the resulting equation of motion for the r^{th} mode is,

$$\ddot{\delta}_r + 2\zeta_r\omega_r\dot{\delta}_r + \omega_r^2\delta_r = \frac{F_r\sin(\Omega t)}{M_r}. \tag{4.17}$$

The solution of Equation 4.17 results in the following displacement,

$$\delta_r(t) = \frac{\frac{F_r}{K_r}}{\sqrt{(1-r_r^2)^2 + (2\zeta_r r_r)^2}}\sin(\Omega t - \alpha) + e^{-\zeta_r\omega_r t}(A_1\cos\omega_d t + \sin\omega_d t) \tag{4.18}$$

where $r_r = \frac{\Omega}{\omega_r}$ is called the frequency ratio, α = phase lag (the angle that the displacement of the system lags the applied force), $\omega_d = \omega_r\sqrt{1-\zeta_r^2}$, is the damped natural frequency and A_1 and A_2 are constants that depend on the initial conditions. The first term on the right side of the last equation corresponds to the steady-state displacement and the second term corresponds to the transient displacement. Substituting the solutions for all the principal DOF into Equation 4.4 leads to the total response over time, including the transient and steady-state displacements, of the original system.

This procedure yields the displacements for only one forcing frequency. It is more convenient and useful to determine the steady-state amplitude of the displacements for a large range of forcing frequencies. This is done by finding the modulus of the frequency response function,

$$|H(\Omega)| = \frac{1}{\sqrt{(1-r_r^2)^2 + (2\zeta_r r_r)^2}} \tag{4.19}$$

which gives the magnitude of the steady-state motion as a function of the frequency ratio r.

Chapter 5

Numerical Results

This chapter demonstrates the capabilities of the computer program DROPS (Dynamic Response of Pole Structures), through a numerical example. Unfortunately, there is no field data available to verify the free vibration or the forced vibration analyses. Due to the size of the problem and complexity of the analysis, hand calculations have not been performed to verify results. However, the three programs written to perform the analysis, see Apendix A, were verified individually with smaller numerical examples, refer to Appendix B for a summary of these examples.

5.1 Description of the Structure

The structure chosen to demonstrate DROPS is a single davit lighting pole. The mounting height of the pole is 19.8 m (65') and the davit radius is 1.8 m (6'), see Figure 5.1 for additional dimensions. The pole consists of two separate sections, a top section made of 7 ga. (3.04 mm) steel and a bottom section made of 11 ga (4.55 mm) steel. A cross section of the pole, which is shown in Figure 5.1, is a dodecagonal (12 sided polygon). The dimension b indicates the nominal length of one side of the dodecagonal. The luminaire, which is not shown in Figure 5.1, is attached to the tip of the pole. The luminaire has a mass of 39 kg and it is added to node 4 in the manner described in Chapter 3.

5.2 Finite Element Model

This section summarizes the procedure and data required by the DROPS program to build the model of the structure. Refer to Appendix A for a description of how to build a model using the DROPS program.

The structure is defined by the four *Defining Nodes* and three *Defining Elements* shown on the left of Figure 5.1. The required material properties for the steel pole are reasonably assumed to be: modulus of elasticity E = 200 GPa, shear modulus G = 80 GPa and mass density ρ = 7850 kg/m^3. The remaining data required to build the model is summarized in Tables 5.1 and 5.2.

Node	X (mm)	Y (mm)
1	0.0	0.0
2	0.0	10972.8
3	0.0	17983.2
4	1828.8	19812.0

Table 5.1: Summary of node locations for example structure.

After the *auto generate* procedure was performed by DROPS the finite element model, shown on the right of Figure 5.1, was defined by 29 nodes, one of which was a support (fixed) node, and 28 elements. This results in a finite element system which has 84 DOF which are not restrained.

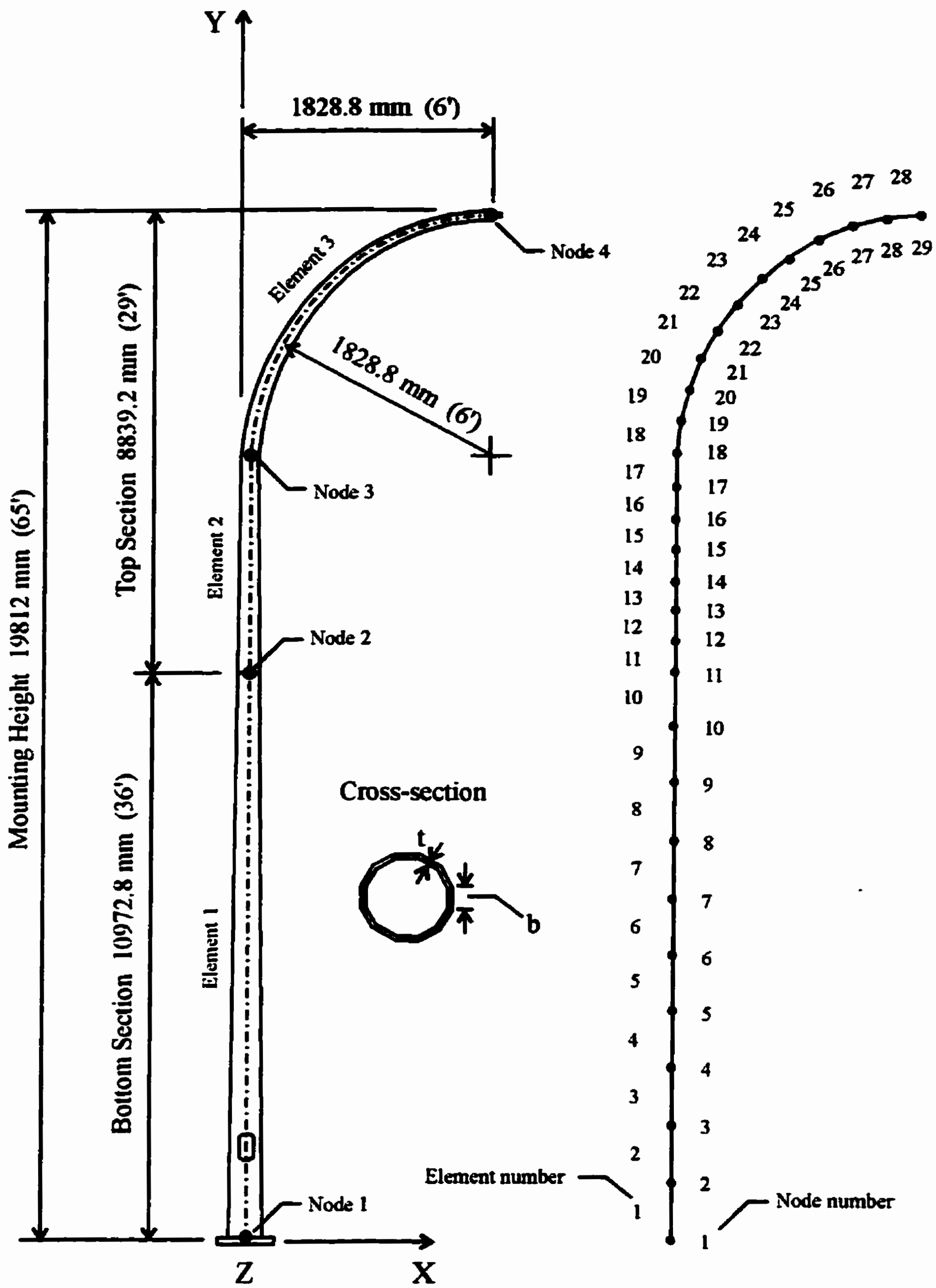

Figure 5.1: Single davit lighting pole.

Element	1	2	3
Curvature (mm)	Straight	Straight	1828.8
Start Node	1	2	3
Start Node t (mm)	4.55	3.04	3.04
Start Node b (mm)	72.1	43.7	25.5
End Node	2	3	4
End Node t (mm)	4.55	3.04	3.04
End Node b (mm)	43.7	25.5	18.1

Table 5.2: Summary of element dimensions for example structure.

5.3 Free Vibration Analysis

From the free vibration analysis, performed by the DROPS program, 84 natural frequencies, one for each unrestrained DOF, and 84 associated mode shapes were determined. Figure 5.2 shows the six lowest natural frequencies and associated mode shapes.

5.4 Forced Vibration Analysis

As mentioned previously, this study focuses on two types of wind loads, gust winds and laminar winds. This section discusses the simulated winds and the resulting response of the example structure.

5.4.1 Gust Wind

The gust wind data which was used in this example is shown in Figure 5.3. The wind data is based on the half cycle of a sine curve, with an amplitude of 5 m/s and a frequency of 0.5 Hz or 3.14 rad/s. This wind speed data was chosen to simulate a

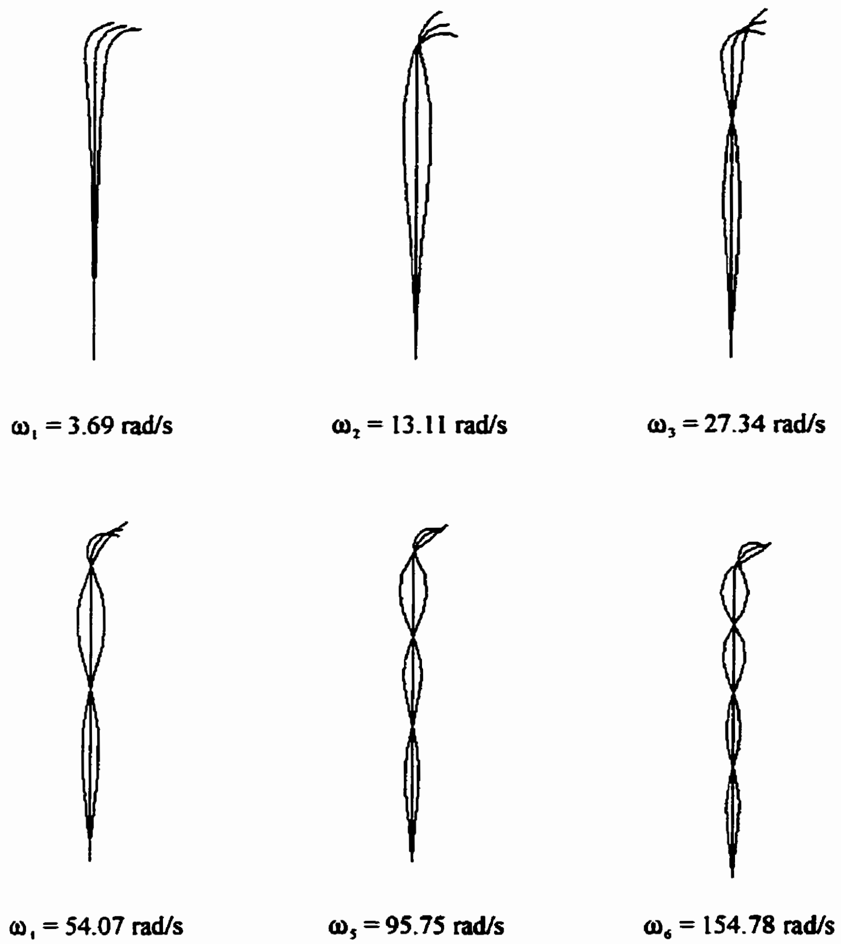

Figure 5.2: Mode shapes for the six lowest natural frequencies.

short pulse of wind. The positive wind direction is in the positive X direction. The wind data is constant over the height of the pole. The first three modes were used in the modal analysis with a modal damping factor of 0.001.

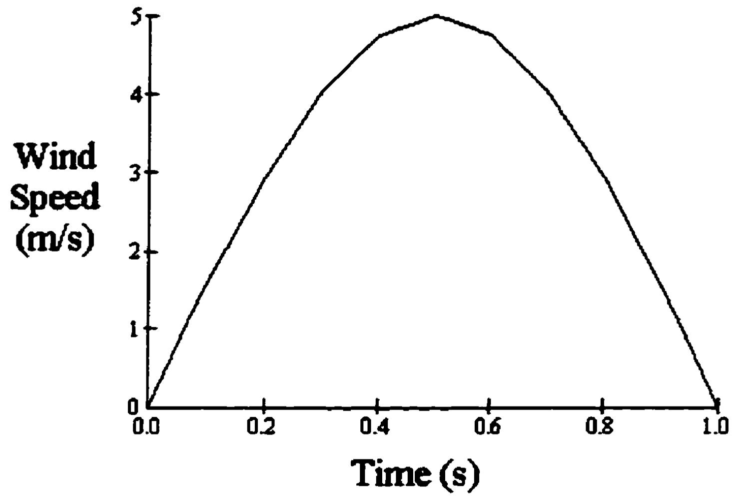

Figure 5.3: Plot of wind speed versus time.

Figure 5.4 shows a temporal plot of the horizontal displacement of the tip of the structure (node 4). The response is as expected. As the wind gust is applied the tip of the structure is displaced in the direction of the wind to a maximum of approximately 15 mm. As the gust diminishes to zero so does the tip displacement. The displacement then continues in the negative direction to a maximum less than 15 mm. This alternating displacement continues well after the gust has diminished. However, the amplitude of the displacement continues to diminish to due to the damping of the structure. If the plot were to continue this amplitude would diminish asymptotically to zero.

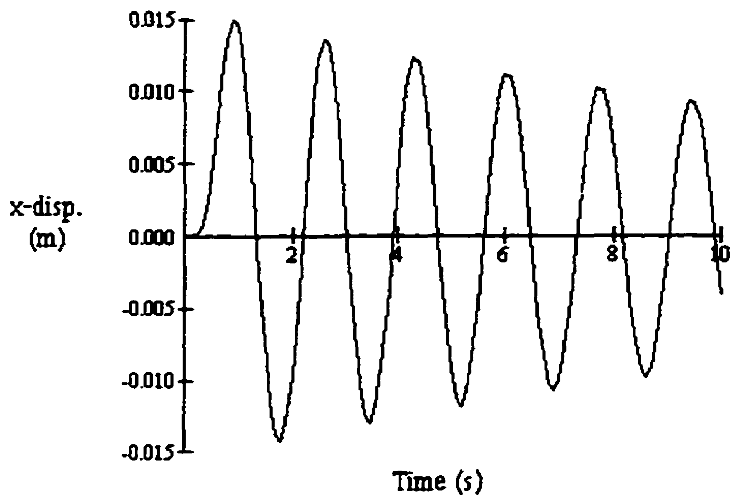

Figure 5.4: Plot of horizontal tip displacement versus time.

5.4.2 Laminar Wind

It has been discussed previously that laminar wind speeds up to 10 m/s are said to cause vortex shedding. Therefore, the laminar wind speed range over which the structure was tested was between 0.1 and 10 m/s. The laminar wind speed is constant over the height of the pole. The first six modes were used in the modal analysis with a modal damping factor of 0.001.

Figure 5.5 shows a plot of the maximum steady-state horizontal tip displacement of the structure versus the laminar wind speed. As can be seen from the plot, the maximum displacement is small over the selected range of wind speeds. However, for certain wind speed the maximum displacement is much larger, indicated by a *spike* in the plot. This wind speed is called a *critical* wind speed. At this wind speed the associated vortex shedding frequency is at or near one of the natural frequencies of the structure resulting in a situation known as resonance. There are a total of six *critical* wind speeds in this range of wind speeds, corresponding to the six lowest natural frequencies discussed previously. The critical wind speeds and associated maximum displacements are sumarized in Table 5.3.

Critical Wind Speed (m/s)	Maximum Displacement (mm)
0.25	9.0
0.75	2.5
1.5	2.0
3.0	2.5
5.5	17.5
8.5	2.0

Table 5.3: Summary of critical wind speeds and associated maximum displacements.

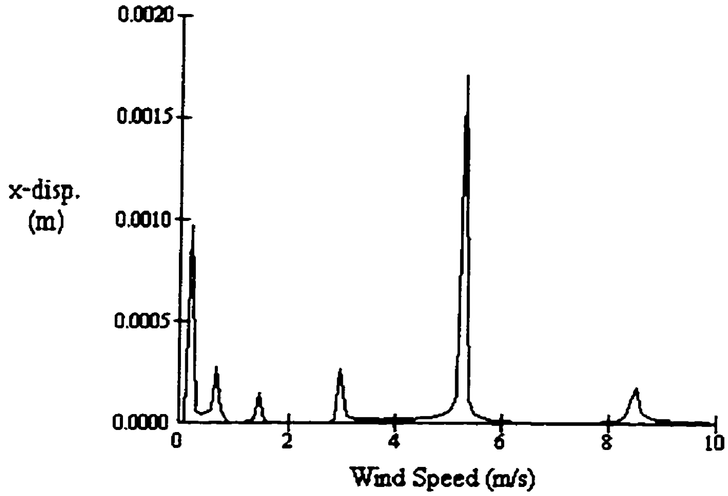

Figure 5.5: Plot of horizontal tip displacement versus laminar wind speed.

Chapter 6

Conclusion

6.1 Concluding Remarks

The purpose of this report was to create a user friendly, interactive computer program, which provides a better understanding of the behavior of street light structures due to wind induced vibrations. The wind conditions and resulting responses investigated corresponds to, low speed laminar winds causing vortex shedding, and gust winds causing vibrations in the direction of the wind. The results determined by the program correspond well with published and hand calculated results.

Critical laminar wind speeds along with the steady-state response due to gust winds are shown graphically in the interactive computer program. The results of individual and specific cases can be studied as the model allows for variation of many parameters including structure geometry, material properties and variation of wind flow characteristics such as velocity, direction and gust wind data. Through the use of the interactive computer program, an in depth understanding of the street light poles currently in use by Manitoba Hydro can be obtained. This may aid in the future design, maintenance and monitoring of these structures, ultimately making their use more efficient and extending their life expectancy.

6.2 Future Work

1. The finite elements chosen to model the pole structure were two noded straight prismatic elements. Using this type of element to model a tapered pole results in a stepped structure for which each successive element has cross sectional properties which are incrementally larger or smaller than the previous element. Another short coming of this type of element is seen in modeling a curved portion of a pole, where the curved portion is modeled by small straight elements. The mathematical model could be improved by creating an element which addresses both of these concerns. An element such as a three noded non-prismatic curved element could be implemented into the mathematical model. This element could be incorporated into the Fortran program, Static, see Appendix A, in the form of an additional element subroutine.

2. An investigation into other forms of wind induced vibrations may also be done and integrated into DROPS. It has been mentioned that galloping may cause perpendicular vibration of pole structures.

3. Full scale field tests which could gain field data for an actual pole structure could be done to further verify and calibrate the computer program.

4. Tow tank and wind tunnel testing could be performed to gain a better understanding of the forces exerted on the structure.

REFERENCES

[1] McDonald, J. R., Mehta, K. C., Oler, W. W. and Pulipaka, N., *Wind Load Effects on Signs, Luminaires and Traffic Signal Structures*, Texas Tech University, 1995.

[2] Krauthammer, T., *A Numerical Study of Wind-Induced Tower Vibrations*, Computers & Structures, Vol. 26, 1987.

[3] Krauthammer, T., Rowekamp, P. A., Leon, R. T., *Experimental Assessment of Wind-Induced Vibrations*, Journal of Engineering Mechanics, Vol 113, 1987.

[4] Kwok, K. C. S., Hancock, G. J., Bailey, P. A., *Dynamics of a Freestanding Steel Lighting Tower*, Engng Struct., Vol. 7, 1985.

[5] Ahmad, M. B., Pande, P. K., Krishna, P., *Self-Supporting Towers Under Wind Loads*, Journal of Structural Engineering, Vol 110, 1984.

[6] Ross, H. E., Edwards, T. C., *Wind Induced Vibration in Light Poles*, Journal of the Structural Division, ASCE, June 1970.

[7] Blevins, R. D., *Flow-Induced Vibration*, Second Edition, Van Nostrand Reinhold, New York, 1990.

[8] Simiu, E. and Scanlan, R. H., *Wind Effects on Structures*, Second Edition, John Wiley and Sons, New York, 1986.

[9] Sachs, P., *Wind Forces in Engineering*, Pergamon Press, Toronto, 1972.

[10] Janna, W. S., *Introduction to Fluid Mechanics*, Wadsworth, Inc., California, 1972.

[11] Bathe, K., *Finite Element Procedures*, Prentice Hall, New Jersey, 1996.

[12] Logan, D. L., *A First Course in the Finite Element Method*, Second Edition PWS-KENT, Boston, 1992.

[13] Gere, J. M., Timoshenko, S. P., *Mechanics of Materials*, Fourth Edition, PWS, Boston, 1997.

[14] Chopra, A. K., *Dynamics of Structures*, Prentice Hall, New Jersey, 1995.

[15] Clough, R. W., Penzien, J., *Dynamics of Structures*, Second Edition, McGraw-Hill, New York, 1993.

[16] James, M. L., Smith, G. M., Wolford, J. C. and Whaley, P. W., *Vibration of Mechanical and Structural Systems with Microcomputer Applications*, Harper Collins, 1993.

[17] Craig, R. Jr., *Structural Dynamics, An Introduction to Computer Methods*, John Wiley and Sons, New York, 1981.

[18] Manitoba Hydro, *Manitoba Hydro Specification No. 17-46M*

Appendix A

Computer Program (DROPS)

A.1 Program Description

The computer program called DROPS (Dynamic Response of Pole Structures), is an interactive Microsoft Windows 95 based program which determines the response of a pole structure due to gust wind and vortex shedding excitations. The program consists of a main interactive program and three separate analysis programs.

The main program was created using Microsoft Visual Basic 4.0, which is a commercial software package used to create Windows based programs. The main program controls the graphical display and user interaction as well as controlling the three analysis programs, Static, Free and Forced. The analysis programs were created by using Microsoft Fortran Power Station, which is a commercial software package used to create Fortran 90 programs. The Static program assembles the finite element model of the structure, the Free program determines the natural frequencies and mode shapes of the model and the Forced program determines the response of the model due to the excitation force.

A.2 Installing DROPS

A.2.1 Installation Requirements

In order to run DROPS, your computer system needs to have the following minimum configurations:

- An IBM or compatible computer capable of running Windows 95 or NT.
- Windows 95 or NT operating system installed on your computer.
- At least 1.2 MB available hard disk space.
- Any Windows-supported monitor and graphics card.
- A mouse is highly recommended.

A.2.2 Installation

1. Start Windows.
2. Insert the DROPS *Disk 1* into the disk drive you are using to install the program.
3. Select *Run* from the *Start* button.
4. If you are using *Drive a*, enter *a:setup* and click *OK*. If you are using a different drive, use that drive letter instead. The installation program will prompt you on screen for the remainder of the installation. The program and required files should be contained in directory *c:\ProgramFiles\Drops*.

A.3 Starting DROPS

1. Start Windows.
2. Make sure that DROPS has been installed.
3. Select *Programs* from the *Start* button. A list of programs should appear on the screen. From this list select the DROPS program. If you wish to make a shortcut to the program or relocate the program on the program list consult Windows 95 Help.

A.4 Building a Model

The procedure for building a finite element model is time consuming and requires an understanding of the finite element method. Refer to Chapter 3 for more information. The DROPS program attempts to simplify this procedure in order to make the program easier to use.

The usual procedure for building a finite element model requires the user to discretize the model into a series of nodes and elements. The nodal coordinates and element connectivity must then be determined for each node and element, respectively. The element properties and material properties for each element must then be determined. For a simple tapered pole this requires numerous hand calculations and a good understanding of the finite element method. The DROPS program reduces this procedure by performing most of these calculations internally by incorporating an *auto generate* procedure.

The *auto generate* procedure requires the user to define nodes only at points of interaction or interest. For example, if a tapered circular pole has a tapered oval luminaire arm attached to it, the user would need only to define the nodes at the ends of each section and the point at which the two are connected. If the connecting point is at the tip of the circular pole and the base of the oval luminaire arm, only three nodes and two elements would need to be defined. These nodes and elements are referred to as *Defining Nodes* and *Defining Elements*, respectively. Once the nodal coordinates, element properties and material properties are defined for the three nodes and two elements, the finite element model can then be *auto generated*. This procedure subdivides each element into a number of elements of equal length and extrapolates the required information for each node and element that is generated. The number of equal length elements can either be specified by the user or automatically determined by the program. The computer program uses a maximum element length of 500 mm for straight sections and 125 mm for curved sections, to determine the number of equal length elements used for the auto generation procedure.

The steps required to build a model using the DROPS program will now be described.

1. Start DROPS program, as described previously. The initial form which is shown should resemble that shown in Figure A.1.

2. From the *File* menu, select *New Model*. You should see the form shown in Figure A.2. Enter the required data and then select the *Ok* button. If you have not entered all the data or the data you have entered is invalid, an error message will appear informing you of the nature of the error. If no error message appears, the form should disappear and a check mark should appear next to *Control Information...* under the *Preprocessor* menu, indicating that this form is complete.

3. Repeat the previous step for *Node Properties...*, *Connectivity...*, *Element Properties...*, *Cross-Sectional Properties...* and *Material Properties...* under the *Preprocessor* menu until all have a check mark next to them.

4. Once all the *Preprocessor* items are complete, you can now generate the finite element model, by selecting *Auto Generate* under the *Preprocessor* menu. This process may take a few seconds to complete depending on your computer.

5. Once the finite element model has been generated from the input data, the finite element model is stored in memory and further analysis of this model may commence.

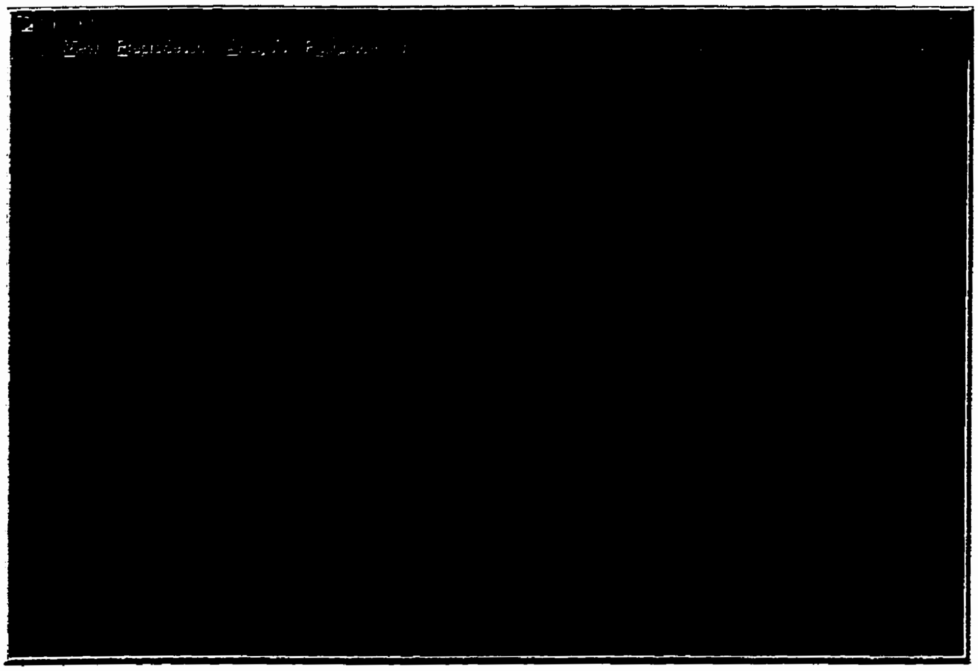

Figure A.1: Main DROPS form.

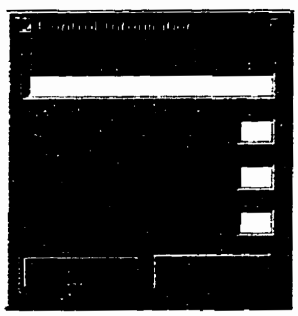

Figure A.2: DROPS Control Information form.

At any point during this process the user can view a graphical image of the model by selecting *Model...* under the *View* menu. The form shown in Figure A.3 should be displayed. By clicking the desired settings on the left of the form the model will be displayed in the area on the right side of the form.

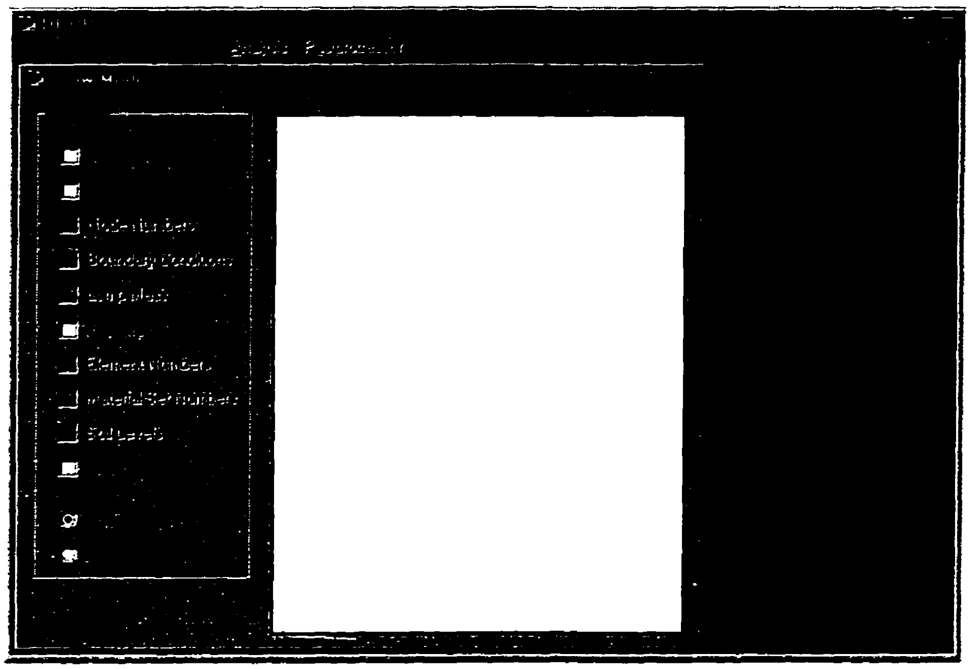

Figure A.3: DROPS Draw Model form.

A.5 Free Vibration Analysis

1. Start DROPS program and build the model as described previously.
2. From the *Analysis* menu select *Free Vibration*. This process may take a few seconds depending on your computer.
3. Once this process has been completed, you can view the natural frequencies and associated mode shapes for the In-Plane (Frame) and Out-Of-Plane (Grid) elements. Refer to Chapter 4 for more information. From the *Postprocessor* menu select *Free Vibration*, a list should appear showing *In-Plane (Frame)...* and *Out-Of-Plane (Grid)...*. By selecting *In-Plane (Frame)...* the form shown in Figure A.4 should appear. A list of the natural frequencies can be found in the drop down menu on the left.

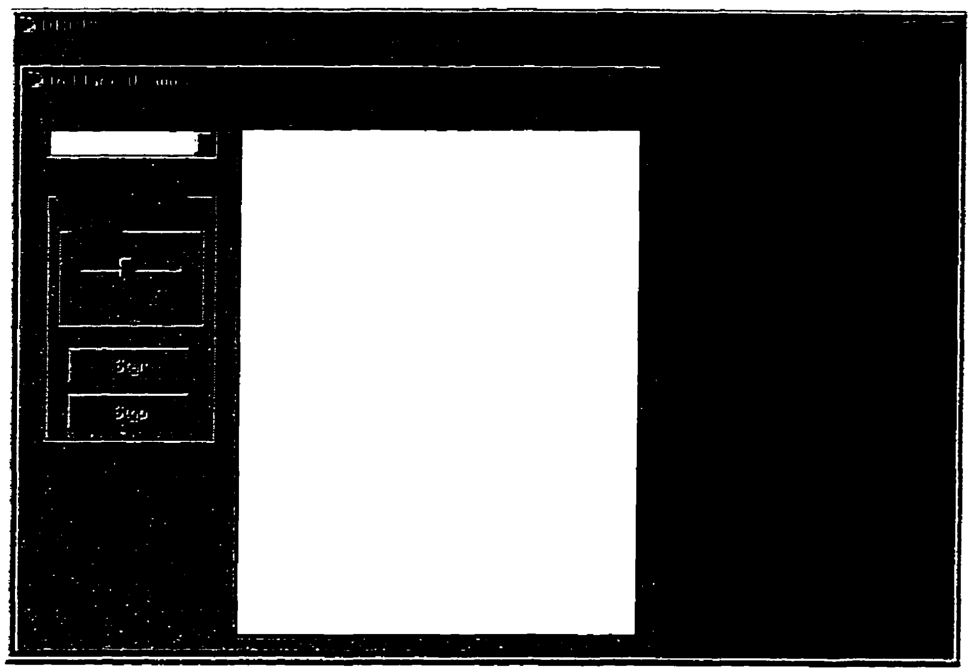

Figure A.4: DROPS In-Plane (Frame) form.

4. By selecting a natural frequency from the list, the associated mode shape will be drawn in the area on the right of the form.
5. To get a better idea of the type of motion associated with the mode shape select the *Start* button on the left side of the form, to begin the *animation* of the mode shape. To stop the animation simply select the *Stop* button on the left side of the form.
6. Once the Free Vibration Analysis is complete the analysis of the Forced Vibration Analysis can be performed.

A.6 Forced Vibration Analysis

The forced vibration analysis is split into *Gust Wind* and *Laminar Wind* cases, which correspond to the along wind response and the response due to vortex shedding. See Chapter 2. We will look separately at how to perform the analysis for each of these wind loading situations.

A.6.1 Gust Wind

1. Start DROPS program, build the model and perform the free vibration analysis as described previously.
2. From the *Analysis* menu select *Forced Vibration.* A list should appear showing *Gust Wind...* and *Laminar Wind....* Select *Gust Wind...* from the list and the form, shown in Figure A.5, should appear. Notice that the menu bar has changed from the Main form shown in Figure A.1.
3. The fist step in this analysis is to create the wind speed data which will be used in the analysis. From the *File* menu select *New Wind Speed Data...*, the form shown in Figure A.6 should appear.
4. Enter the time and corresponding wind speed data, for the wind speed record you wish to use, into the form.
5. When all data has been entered select the *Ok* button. If all data is valid the form will disappear and a check mark should appear next to *Wind Speed Data...* under the *Procedure* menu, indicating that this form is complete.
6. The next step is to specify the wind direction. From the *Procedure* menu select *Wind Direction...* and the form shown in Figure A.7 should appear.
7. To specify the positive wind direction, simply select one of the four gray arrows. Notice when you select the gray arrow it should turn to a blue arrow indicating the direction you have selected. To change the wind direction simply select one of the other three gray arrows.

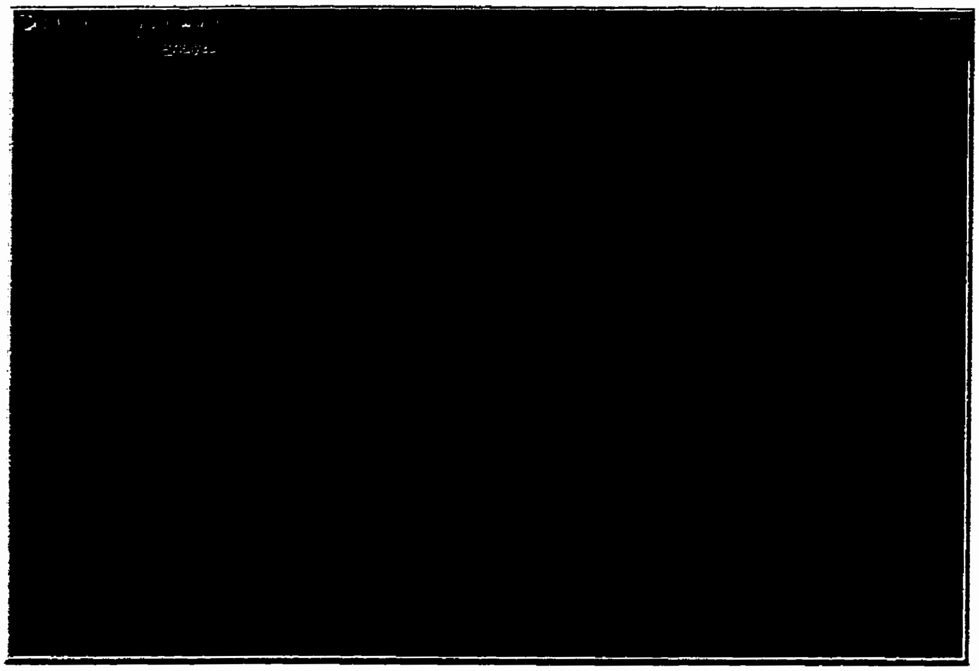

Figure A.5: DROPS - [Gust Wind] form.

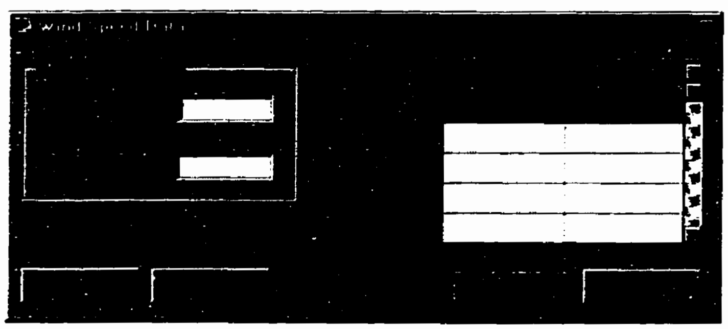

Figure A.6: DROPS - [Gust Wind], Wind Speed Data form.

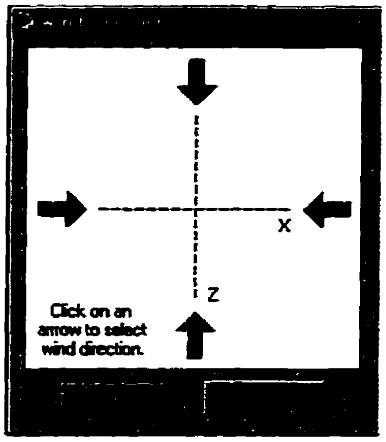

Figure A.7: DROPS Wind Direction form.

8. Once you have specified the desired wind direction, select the *Ok* button. If there are no errors the form should disappear and a check mark should appear next to *Wind Direction...* under the *Procedure* menu, indicating that this form is complete.

9. You may perform the analysis by selecting *Analyze* from the menu. This process may take some time to complete, depending on your computer and the size of the problem.

10. Once this process has been completed, you can view the response of the model by selecting *Forced Vibration* under the *Postprocessor* menu. The form shown in Figure A.8 should appear. To view the response at a specific node and specific DOF, simply select the desired node from the drop down list and the desired DOF from the list of available DOF. The temporal response of the DOF should appear graphically on the right side of the form.

A.6.2 Laminar Wind

1. Start DROPS program, build the model and perform the free vibration analysis as described previously.

2. From the *Analysis* menu select *Forced Vibration*. A list should appear showing *Gust Wind...* and *Laminar Wind....* Select *Laminar Wind...* from the list and the form, shown in Figure A.9, should appear.

3. The fist step in this analysis is to define the laminar wind speed range. From the *Procedure* menu select *Wind Speed Range...*, the form shown in Figure A.10 should appear. Enter the desired low and high range of wind speeds on the form

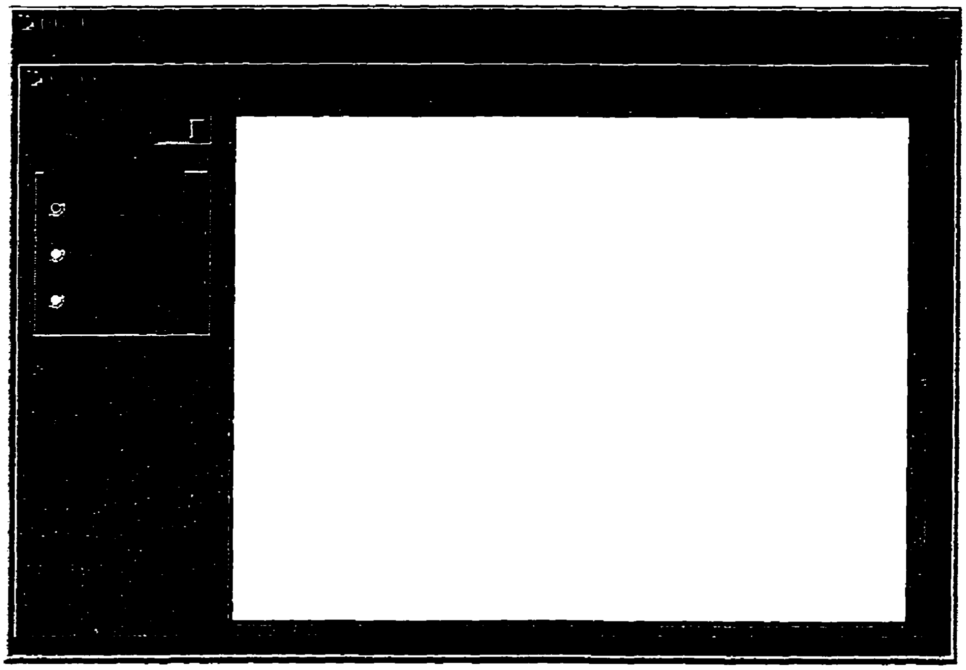

Figure A.8: DROPS Post Processor form.

and select the *Ok* button. If there are no errors the form should disappear and a check mark should appear next to *Wind Speed Range...* under the *Procedure* menu, indicating that this form is complete.

4. Define the wind direction, as described previously, if it has not yet been defined.
5. You may perform the analysis by selecting *Analyze* from the menu. This process may take some time to complete, depending on your computer and the size of the problem.
6. Once this process has been completed, you can view the response of the model by selecting *Forced Vibration* under the *Postprocessor* menu. The form shown in Figure A.8 should appear. To view the response at a specific node and specific DOF, simply select the desired node from the drop down list and the desired DOF from the list of available DOF. The response of the DOF versus wind speed should appear on the right side of the form.

A.6.3 Analysis Options

Additional control over the analysis procedure may be attained by the user through the analysis options form. The following steps will describe the procedure to change the analysis options which are originally set to default values.

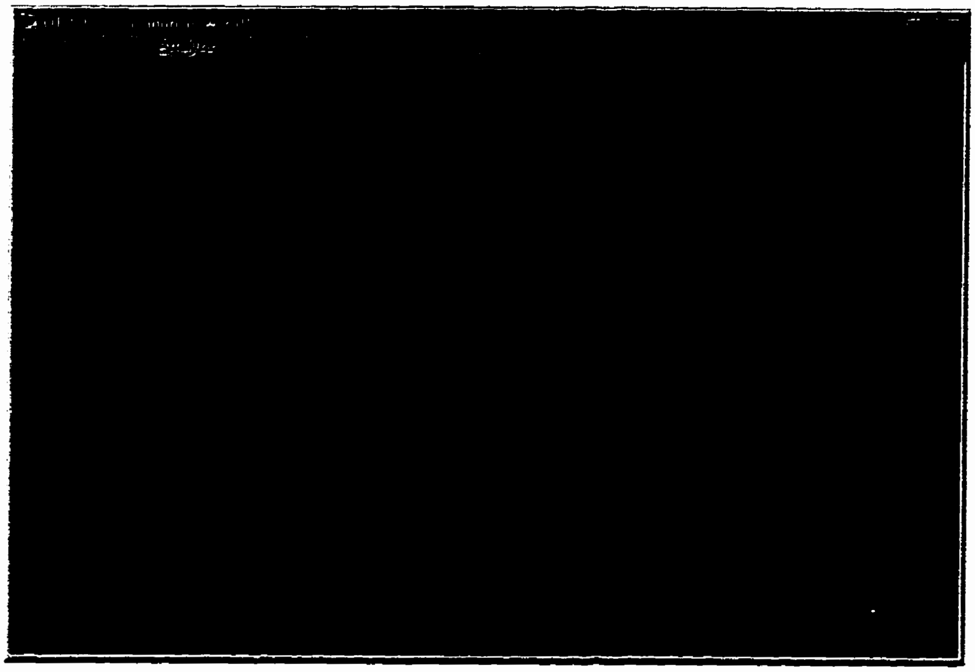

Figure A.9: DROPS - [Laminar Wind] form.

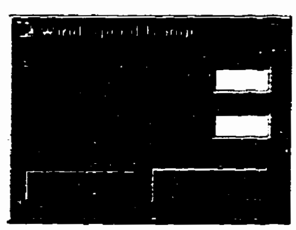

Figure A.10: DROPS Wind Speed Range form.

1. Follow the steps described previously for the Gust Wind or Laminar Wind up to the step prior to selecting the *Analyze* menu.

2. From the *Procedure* menu select *Analysis Options...*, the form shown in Figure A.11 should appear. The options available are different depending on which type of wind load is being considered. Shown in Figure A.11 are the options available for the *Gust Wind* analysis. The first two options control the modal analysis procedure and are common to both wind loads. However, the third is available only for the gust wind and controls the total time for which the time integration is to be performed.

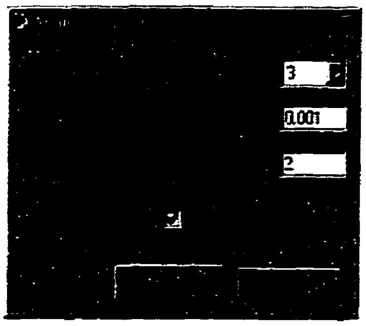

Figure A.11: DROPS Analysis Options form.

3. To change the options first select the check box labeled *Use Default Values*. This should allow you to change the available options. By again selecting the check box the options will be reset to the default values.

4. Once you have entered the desired options select the *Ok* button. If there are no errors, the form will disappear and you may continue with the analysis as described previously.

A.7 Material Properties Database

The following steps will show you how to add more material types to the list found in the Material Properties form.

1. Locate the file *Mat.db* which should be located in the same directory as the DROPS program *c:\ProgramFiles\Drops*. Open this file in any text editor such as *Microsoft Notepad*. The first line of the file indicates the total number of material types in the database. Each material type in the database consists of a material name, maximum of twenty characters long, the modulus of elasticity in GPa, the shear modulus in GPa and the mass density in kg/m^3. Each of the material properties are entered on a new line of the database.

2. To create a new material type in the database, simply add the four required properties to the end of the file making sure that each property is on a new line.
3. Once you have added one or more new material types to the database make sure to change the number at the top of the list to indicate the total number of material types in the modified database.
4. Save the file and exit the text editor.

Appendix B

Verification of the Program

This appendix summarizes the numerical examples which were used to verify the analysis programs Static, Free and Forced described in Appendix A.

B.1 Example 1

The purpose of the Static program is to assemble the mass and stiffness matrices of the finite element model. A 2 m tapered steel pole, shown in Figure B.1, was used to verify this aspect. The pole was modeled by using two prismatic frame elements which resulted in a problem which was small enough to verify by hand calculations but large enough to fully test the program. The cross section of the pole was assumed to be circular with an outside diameter at the base of 200 mm and 100 mm at the tip. The thickness of the steel was assumed invariably to be 3 mm. The modulus of elasticity and mass density were assumed to be 200 GPa and 7850 kg/m^3, respectively.

The resulting stiffness and mass matrices are shown in Equations B.1 and B.2. The matrices, which were determined by hand calculations, resulted in a maximum discrepancy of 0.04%.

$$[K] = \begin{bmatrix} 19531200 & 0 & 4627200 & -5138400 & 0 & -2569200 \\ 0 & 5.54 \times 10^8 & 0 & 0 & -2.30 \times 10^8 & 0 \\ 4627200 & 0 & 6510400 & 2569200 & 0 & 856400 \\ -5138400 & 0 & 2569200 & 5138400 & 0 & 2569200 \\ 0 & -2.30 \times 10^8 & 0 & 0 & 2.30 \times 10^8 & 0 \\ -2569200 & 0 & 856400 & 2569200 & 0 & 1712800 \end{bmatrix} \tag{B.1}$$

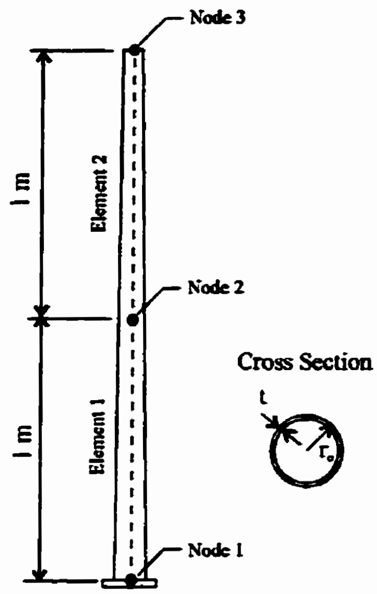

Figure B.1: Simple tapered pole of Example 1.

$$[M] = \begin{bmatrix} 8.079 & 0 & 0.1937 & 1.161 & 0 & 0.2794 \\ 0 & 7.251 & 0 & 0 & 1.505 & 0 \\ 0.1937 & 0 & 0.2072 & -0.2794 & 0 & -0.0645 \\ 1.161 & 0 & -0.2794 & 3.353 & 0 & 0.4729 \\ 0 & 1.505 & 0 & 0 & 3.001 & 0 \\ 0.2794 & 0 & -0.0645 & 0.4729 & 0 & 0.0860 \end{bmatrix} \tag{B.2}$$

B.2 Example 2

The purpose of the Free program is to find the natural frequencies and mode shapes of the model assembled by the Static program. The example chosen to verify this program was an example taken from a vibrations text book [16].

The frame shown in Figure B.2 is fixed at nodes 1 and 4 and is restrained in the vertical direction at nodes 2, 3, 5 and 6. The required element and material properties are; modulus of elasticity $E = 30 \times 10^6$ psi, cross sectional area $A = 17.634$ in^2, moment of inertia $I = 984$ in^4 and mass density $\rho = 7.372 \times 10^{-4}$ lb/in^4.

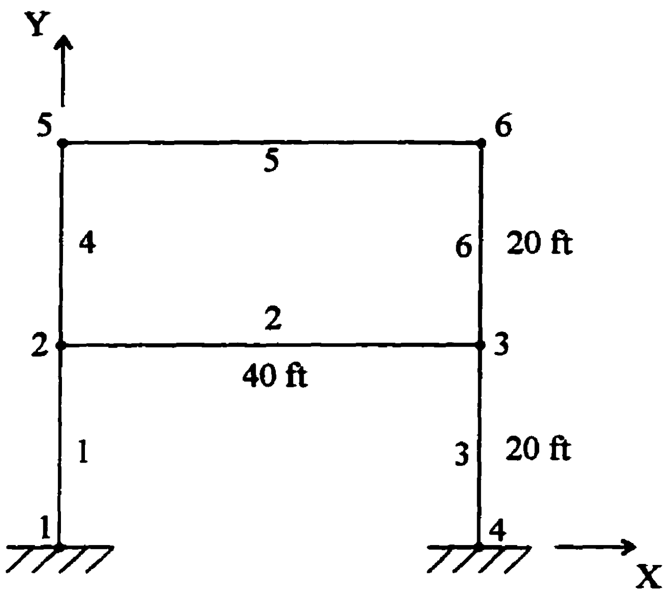

Figure B.2: Frame of Example 2.

The first three natural frequencies are $\omega_1 = 27.62$ rad/s, $\omega_2 = 97.44$ rad/s and $\omega_3 = 137.48$ rad/s. Equation B.3 shows the associated mode shapes for the first three natural frequencies. These results have a maximum discrepancy of 0.09% when compared to those reported in the vibrations text book.

$$\phi_1 = \begin{Bmatrix} 1.0 \\ -0.00441 \\ 1.0 \\ -0.00441 \\ 2.142 \\ -0.00273 \\ 2.142 \\ -0.00273 \end{Bmatrix}, \phi_2 = \begin{Bmatrix} 1.0 \\ 0.00057 \\ 1.0 \\ 0.00057 \\ -0.0712 \\ 0.00624 \\ -0.0712 \\ 0.00624 \end{Bmatrix}, \phi_3 = \begin{Bmatrix} 1.0 \\ -0.286 \\ -1.0 \\ 0.286 \\ -0.193 \\ 0.626 \\ 0.193 \\ -0.626 \end{Bmatrix} \tag{B.3}$$

B.3 Example 3

This example verifies the analysis of a system due to a general time dependent excitation, performed by the Forced program. The analysis is done as described in Chapter 4. The example chosen to verify this program was an example from a vibration text book [16].

A five-story building was modeled by the system shown in Figure B.3. The system consists of five elements having stiffness values of; $k_1 = k_2 = 10 \times 10^7$ lb/in, $k_3 = k_4 = 8 \times 10^7$ lb/in and $k_4 = 6 \times 10^7$ lb/in. The mass of the structure is modeled by five lump masses added at each node, the values of the lump masses are; $m_1 = m_2 = m_3 = 65 \times 10^3$ lb s^2/in, $m_4 = 60 \times 10^3$ lb s^2/in and $m_5 = 45 \times 10^3$ lb s^2/in. The building was said to be exposed to tornado wind loading which was modeled by applying a forcing function, $F(t)$, at each floor. The data used for the forcing function is summarized in Table B.1.

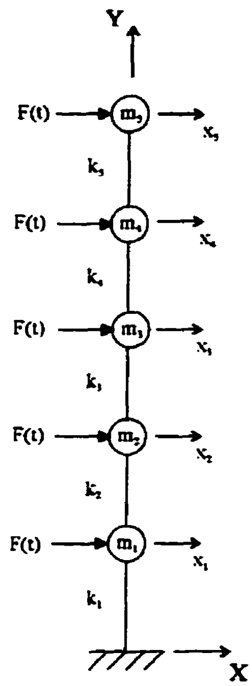

Figure B.3: Model of five-story building used in Example 3.

The part of the solution from the time integration is summarized in Table B.2. This solution matches the solution reported in the text book, with a maximum discrepancy of 0.05%.

time, s	F(t), lb
0.0	0.0
0.20	4.0×10^7
0.35	4.8×10^7
0.60	5.0×10^7
0.90	4.6×10^7
1.02	4.0×10^7
1.05	3.0×10^7
1.08	2.0×10^7
1.13	1.0×10^7
1.20	0.0
1.30	-1.0×10^7
1.40	-1.7×10^7
1.45	-2.0×10^7
1.65	-2.6×10^7
2.00	-3.0×10^7
2.15	-2.7×10^7
2.30	-2.0×10^7
2.45	-1.0×10^7
2.50	0.0

Table B.1: Summary of tornado wind loading.

Time, s	x_1, in	x_2, in	x_3, in	x_4, in	x_5, in
0.0	0.000	0.000	0.000	0.000	0.000
0.01	0.000	0.001	0.001	0.001	0.001
0.02	0.003	0.005	0.004	0.004	0.006
0.03	0.011	0.015	0.014	0.015	0.020
0.04	0.025	0.036	0.033	0.035	0.047
0.05	0.047	0.068	0.064	0.069	0.090
0.06	0.078	0.114	0.112	0.120	0.153
0.07	0.118	0.175	0.179	0.193	0.239
0.08	0.166	0.253	0.268	0.291	0.350
0.09	0.222	0.346	0.382	0.417	0.490
0.10	0.286	0.456	0.522	0.574	0.660
⋮	⋮	⋮	⋮	⋮	⋮
0.38	3.645	6.759	9.687	11.661	12.974
0.39	3.656	6.784	9.731	11.713	13.025
0.40	3.648	6.776	9.729	11.710	13.010
0.41	3.622	6.735	9.681	11.650	12.932
0.42	3.580	6.664	9.585	11.533	12.794

Table B.2: Summary solution for Example 3.

B.4 Example 4

This example verifies the analysis of a system due to harmonic excitation, performed by the Forced program. The analysis is done as described in Chapter 4. The example chosen to verify this program was an example from a vibration text book [17].

A four-story building was modeled by the system shown in Figure B.4. The system consists of four elements having stiffness values of; $k_1 = 800$ kips/in, $k_2 = 1600$ kips/in, $k_3 = 2400$ kips/in and $k_4 = 3200$ kips/in. The mass of the structure is modeled by four lump masses added at each node, the values of the lump masses are; $m_1 = 1$ kips s^2/in, $m_2 = 2$ kips s^2/in, $m_3 = 2$ kips s^2/in and $m_4 = 3$ kips s^2/in. The building was excited by applying a harmonic load at the top floor equal to $\cos \Omega t$.

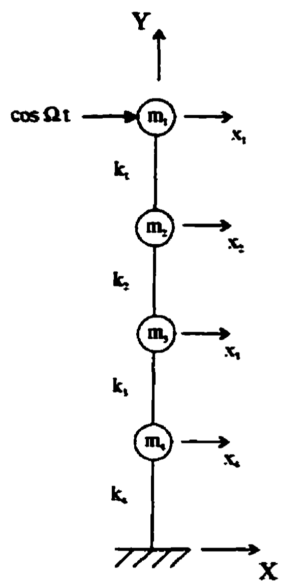

Figure B.4: Model of four-story building used in Example 4.

The maximum displacement of the system at node 1 is summarized in Table B.3. The response found by the Forced program have a maximum discrepancy of 0.04% when compared to those reported by the text book.

Forcing Frequency Ω, rad/s	Amplitude of $x_1(t)$ in
0	0.002602
6.647	0.003288
53.403	-0.000499

Table B.3: Summary solution for Example 4.

Appendix C

Nomenclature

Re = Reynolds number

V = free stream wind velocity

d = bluff diameter

ν = kinematic viscosity of air

S = Strouhal number

f_s = vortex shedding frequency

A = area upon which the wind force acts

ρ_{air} = mass density of air

F_D = drag force

F_L = lift force

ω_s = circular vortex shedding frequency

L = length of an element

θ = angle of inclination to the wind

FEM = Finite Element Method

DOF = Degree(s)-Of-Freedom

X, Y, Z = global coordinate system

x, y, z = local coordinate system

u, v, w = translational DOF in the local coordinate system

d_X, d_Y, d_Z = translational DOF in the global coordinate system

ϕ_x, ϕ_y, ϕ_z = rotational DOF in the local coordinate system

ϕ_X, ϕ_Y, ϕ_Z = rotational DOF in the global coordinate system

$[k_e]$ = elemental local stiffness matrix

$[m_e]$ = elemental local mass matrix

E = modulus of elasticity

A = cross sectional area

I = moment of inertia

ρ = mass density

G = shear modulus

J = polar moment of inertia

$[T]$ = transformation matrix

$[K]$ = global stiffness matrix

$[M]$ = global mass matrix

M_L = total mass of additional non-structural component

ω = natural frequencies

ϕ = mode shape

$\{\ddot{X}\}, \{\dot{X}\}, \{X\}$ = global acceleration, velocity and displacement vectors

$[C]$ = global damping matrix

$\{F(t)\}$ = forcing function vector

$[\Phi]$ = modal matrix

$\{\delta\}$, $\{\dot{\delta}\}$, $\{\ddot{\delta}\}$ = displacement, velocity and acceleration vectors in terms of the principal coordinates

$[\mathbf{M}]$, $[\mathbf{C}]$, $[\mathbf{K}]$ = modal mass, damping and stiffness matrices

$\{\mathbf{F}(t)\}$ = modal force vector

M_r,C_r,K_r = diagonal components of the modal mass, damping and stiffness matrices for the r^{th} mode

$F_r(t)$ = modal forcing function for the r^{th} mode

ζ_r = modal damping factor for the r^{th} mode

$E_r(t)$ = excitation function of the r^{th} mode

Δt = time integration increment

k_1,k_2,k_3,k_4 = time integration constants

Ω = frequency of a harmonic excitation

r = frequency ratio

α = phase lag

ω_d = damped natural frequency

$H(\Omega)$ = frequency response function