# Structural Analysis for Electric Power Transmission Structures

William F. Cofer, Ph. D., PE
Department of Civil and Environmental Engineering
Washington State University

## 1.0 Introduction

The electric power transmission industry is facing numerous changes in the ways in which traditional structures are being designed. De-regulation, and subsequent re-regulation, has resulted in increased oversight of the design process. Concurrently, budget reductions have led to staff reductions, often achieved through retirements and promotions. Thus, there is now increased reliance upon computer software for the performance-based design of electric power transmission structures. At the heart of the software is the structural analysis module, through which the behavior of the structure when subjected to numerous combinations of load is simulated. In this paper, the development and the mathematical basis for the structural analysis module within LD-PRO is described.

### 1.1 Types of Structures

A number of structure types are described in relevant publications (ASCE, 1997; USDA, 1992). They include single poles or masts, H-frames, rigid frames, masted towers, and space trusses. Generally, they are guyed structures and they may be constructed of wood, steel, aluminum, or reinforced or prestressed concrete. Members are often slender, tapered, and subject to axial compression. Thus, deflections and member deformations are often significant and the structures may be subject to instability and material failure.

### 1.2 Types of Loads

Electric power transmission structures must be designed to resist load from self-weight, guy tension, wire forces, wind, ice, temperature, and longitudinal loads due to a broken conductor, differential ice conditions, stringing loads, or a change in ruling span.

### 1.3 Types of Behavior

Traditional linear elastic theory for the analysis of structures includes assumptions that displacements and member deformations are so small that they do not affect interactions between forces, that material behavior is linear, and that there is no change in member behavior during loading. However, the geometric effects from the influence of axial force on flexural stiffness, the destabilizing moment of gravity acting on lateral displacement (i.e., the P-Δ effect), and the change in orientation of the members of the structure as it undergoes relatively large displacements and deformations should be considered in the analysis of electric power transmission structures. In addition, the behavior of guys and the possibility of cracking or yielding in individual members makes the behavior inherently nonlinear for all but the simplest load cases.

Important modes of failure include buckling of poles, cracking/yielding of members, and general instability/collapse of the structure. Other effects that are important but rarely considered include the flexibility of pole and guy supports and creep.

### 1.4 Design Criteria and Codes

The National Electric Safety Code (NESC) is the predominant code used for structural design within the industry. The NESC specifies design values for temperature, ice thickness, and wind pressure in designated loading districts. Load factors are then applied to each effect and combinations of effects are specified for structural analysis to evaluate the capacity of the structure.

## 2.0 Description of the Structural Analysis Module

The purpose of the structural analysis module is to provide a framework for the development of a mathematical model that captures the important aspects of the response of the structure when subjected to combinations of load. Results of the analysis include axial force, shearing force, and bending moment in beam/pole members, axial force in truss/guy members, deflections/rotations, and support reactions.

### 2.1 Introduction to the matrix method of structural analysis

Virtually all modern structural analysis software is based on the direct stiffness method, which is a subset of the Finite Element Method. Here, the structure is considered to be an assemblage of members (or elements) and joints (or nodes). Each node is assumed to have a location in space, defined by coordinates, and a set of degrees of freedom. Degrees of freedom are the components of a displacement vector (designated as $u$, $v$, and $w$ in three dimensional space) and the components of a rotation vector (designated as $q_x$, $q_y$, and $q_z$). With each degree of freedom is a corresponding force or moment that, when multiplied by the degree of freedom, has units of work. Figure 1 shows a structural model with six nodes and five elements. Each node has three coordinates in the global coordinate system, as shown, and six degrees of freedom.

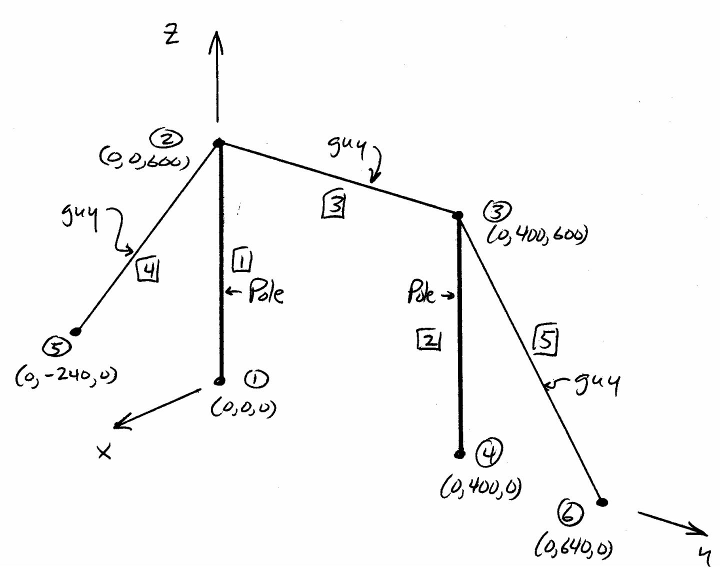
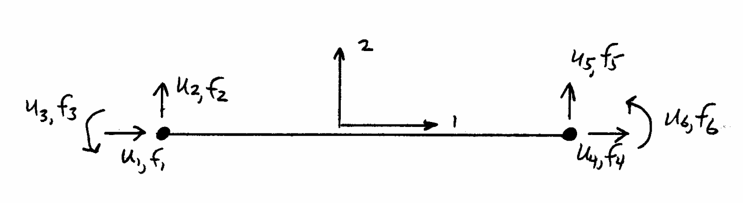


Figure 1. Example of a structural model.

Elements are defined to connect nodes, and each element contributes stiffness that relates forces (or moments) to the corresponding displacement (or rotation) degrees of freedom of the nodes it connects. Because there are multiple degrees of freedom for an element, there are numerous stiffness coefficients. The stiffness coefficients are arranged into a *stiffness matrix* that relates a vector of all degrees of freedom for the nodes of the element to a vector of all corresponding force/moment components for the nodes of the element. Stiffness coefficients are usually given in terms of cross section and material properties of the element. For example, Figure 2 shows the stiffness matrix and the degrees of freedom for a two dimensional frame element. The general form of the equation given is $\underline{f} = \underline{\underline{k}}\,\underline{u}$, which is an equation of equilibrium that states "external applied forces (i.e., $\underline{f}$) are balanced by internal forces (i.e., $\underline{\underline{k}}\,\underline{u}$)". Note that stiffness matrices for structural elements are typically defined in a local frame of reference, defined to run along the length of the element.



$$\begin{Bmatrix} f_1 \\ f_2 \\ f_3 \\ f_4 \\ f_5 \\ f_6 \end{Bmatrix} = \begin{bmatrix} \frac{AE}{L} & 0 & 0 & -\frac{AE}{L} & 0 & 0 \\ 0 & \frac{12EI}{L^3} & \frac{6EI}{L^2} & 0 & -\frac{12EI}{L^3} & \frac{6EI}{L^2} \\ 0 & \frac{6EI}{L^2} & \frac{4EI}{L} & 0 & -\frac{6EI}{L^2} & \frac{2EI}{L} \\ -\frac{AE}{L} & 0 & 0 & \frac{AE}{L} & 0 & 0 \\ 0 & -\frac{12EI}{L^3} & -\frac{6EI}{L^2} & 0 & \frac{12EI}{L^3} & -\frac{6EI}{L^2} \\ 0 & \frac{6EI}{L^2} & \frac{2EI}{L} & 0 & -\frac{6EI}{L^2} & \frac{4EI}{L} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \\ u_5 \\ u_6 \end{Bmatrix}$$

Figure 2. Stiffness matrix for a two dimensional frame element.

Stiffness matrices of a similar form may be defined for many types of elements. Then, a full structural model may be formed by overlaying the stiffness matrices of many elements such that the stiffness coefficients pertain to the proper global degrees of freedom. For example, in the structure shown in Figure 1, the degrees of freedom of node 2 would be affected by the stiffness of elements 4, 1, and 3. An equation of equilibrium for the entire structure is given as:

$$\underline{f}_{system} = \underline{\underline{k}}_{system} \underline{u}_{system} \tag{1}$$

where $\underline{f}_{system}$ and $\underline{u}_{system}$ represent all of the degrees of freedom of the system and $\underline{\underline{k}}_{system}$ is the system stiffness matrix, which contains the stiffness coefficients of all elements. Note that, before being summed, the stiffness matrices of each element must be transformed from the element local frame of reference to the system global frame of reference.

Equation 1 is an equation of equilibrium for the system. Assuming that applied forces, which reside in $\underline{f}_{system}$, are known, the equation must be solved for $\underline{u}_{system}$. However, the system stiffness matrix is singular (i.e., its inverse does not exist) without modification to recognize supports. Supports are degrees of freedom for which the displacement is prescribed, usually to a zero value. Enough degrees of freedom in the system must be prescribed to restrict rigid-body motion. In three dimensions, this means that at least six degrees of freedom must be prescribed in such a way as to prevent three components of rigid-body displacement and three components of rigid-body rotation. For the structure of Figure 1, for example, all six degrees of freedom would be prescribed for nodes 1 and 4 to represent fixity at the base of the poles and the three displacement degrees of freedom would be prescribed at nodes 5 and 6 to represent pin supports at the base of the guys.

With proper support, the system stiffness matrix is made nonsingular and Equation 1 may be solved for $\underline{u}_{system}$. With values known for each degree of freedom in the structure, forces in individual elements may now be found by multiplying appropriate degree of freedom values by the element stiffness matrices. Finally, reaction force values for prescribed degrees of freedom are obtained by summing the element forces from the ends of all elements attached to the node in question.

Often, loads that are applied to the structural model cannot be represented as discrete forces or moments at nodes. Examples include element weight, wind load, ice load, pretension in guys, and temperature effects. In these cases, loads are defined as being applied to elements. Here, basic principles of mechanics are used to compute the reaction forces that would be applied to the ends of the element, when subjected to load, if those ends were assumed to be perfectly fixed. These "fixed-end forces" are then reversed and assumed to contribute to the forces applied to the nodes of the system. After solving for $\underline{u}_{system}$, element forces are presented as the sum of those computed earlier with element load and fixed ends and those computed from end movement, as given by the displacement solution.

## 2.2 P-Δ Analysis

As described earlier, electric power transmission structures are often constructed of slender elements with significant lateral displacement. Gravity loads acting on that lateral displacement cause an increase in moment in that element. This increase in moment is considered to be *second-order*, and it is typically called the *P-D effect.*

One way to include the P-Δ effect would be to reformulate the equations of equilibrium (i.e., $\underline{f}_{system} = \underline{\underline{k}}_{system}\underline{u}_{system}$) with updated geometry in a recursive way. Indeed, this approach is followed when nonlinear analysis is used. However, when the dominant effect on structural behavior is the P-Δ effect, an excellent approximation can be obtained by including the effect of axial force on the flexural stiffness of each element. This is done through the use of the *geometric stiffness matrix*.

The geometric stiffness matrix has its name because it acts like a stiffness matrix, in that it relates the degrees of freedom at each node of an element to their corresponding force components. However, it is called geometric because its coefficients are functions of element length and axial force only. For example, the geometric stiffness matrix for the two dimensional frame element of Figure 2 is given in Figure 3 (McGuire, et al., 2000), where *P* is the element axial force.

$$\begin{Bmatrix} f_1 \\ f_2 \\ f_3 \\ f_4 \\ f_5 \\ f_6 \end{Bmatrix} = \frac{P}{L} \begin{bmatrix} 1 & 0 & 0 & -1 & 0 & 0 \\ 0 & \frac{6}{5} & \frac{L}{10} & 0 & -\frac{6}{5} & \frac{L}{10} \\ 0 & \frac{L}{10} & \frac{2L^2}{15} & 0 & -\frac{L}{10} & -\frac{L^2}{30} \\ -1 & 0 & 0 & 1 & 0 & 0 \\ 0 & -\frac{6}{5} & -\frac{L}{10} & 0 & \frac{6}{5} & -\frac{L}{10} \\ 0 & \frac{L}{10} & -\frac{L^2}{30} & 0 & -\frac{L}{10} & \frac{2L^2}{15} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \\ u_5 \\ u_6 \end{Bmatrix}$$

Figure 3. Geometric stiffness matrix for a two dimensional frame element.

For each element, the geometric stiffness matrix is added to the usual, first-order stiffness matrix to form the *tangent stiffness matrix,* $\underline{\underline{k}}_t$:

$$\underline{\underline{k}}_t = \underline{\underline{k}} + \underline{\underline{k}}_g \tag{2}$$

The tangent stiffness matrix is then used in the same way as the first-order stiffness matrix in linear analysis. Two items must be mentioned, however. The axial force, *P*, is positive when it is tensile. Then, the geometric stiffness matrix serves to increase the value of the tangent stiffness. Conversely, when the axial force is compressive, *P* is negative and the geometric stiffness matrix reduces the tangent stiffness. For a high enough value of compressive axial force, the tangent stiffness matrix is no longer positive definite. For the structure, elastic instability is indicated when the tangent stiffness matrix changes from positive definite to negative definite.

The second item is that, at the beginning of the analysis, axial forces in the elements are unknown. Thus, several iterations may be required before the final result is obtained. For the first iteration, a first-order analysis is performed and element forces are computed. Knowing element axial forces, the analysis is repeated, but tangent stiffness matrices are used. Element forces are again computed and updated, and the analysis is repeated. The procedure continues until changes are small. Typically, only a few iterations are required, and this type of analysis is only slightly more demanding than linear, first-order analysis.

### 2.3 Nonlinear analysis

Although the use of the tangent stiffness matrix enhances the first-order solution, the effects of finite deflections and rotations, material yield/cracking, and tightening/slackening of guys are not included in P-Δ analysis. True second-order analysis requires that the updated geometry of the structure and changes in stiffness from other effects be accounted for in formulating the equations of equilibrium. Several changes in the solution procedure are thus required:

1. Loads are applied in increments. This is accomplished by defining the pattern of loads on the structure, which are then acted upon by a load multiplier, denoted as $l$, that varies incrementally from 0 to 1.
2. For each increment of loading, the structure is updated. This involves relocating the nodes in accordance with deflections, updating the tangent stiffness matrix in accordance with element forces, reformulating the cable equations for guys, and evaluating the section properties of elements to account for cracking or yielding. Here, all measures of stress and strain are defined with respect to the most current configuration, which is known as the Updated Lagrangian approach (Bathe, 1996, Yang and Kuo, 1994).
3. For each load increment, equilibrium must be established before going on. Here, after solving for node displacements for a step of the analysis (often called the *predictor* phase), the system is updated and internal forces are evaluated and compared with applied forces (often called the *equilibrium-checking* phase). If they are out of balance (i.e., the system is out of equilibrium), the residual forces are applied as additional load (often called the *corrector* phase). The process is repeated iteratively until the residual forces and the change in node displacements are small. This is the process of obtaining *convergence*. With convergence comes the assurance that equilibrium is satisfied within a given tolerance, including all nonlinear effects.

The analyst must specify the parameters that define the load increments and the criteria for convergence. The inability to obtain convergence could indicate collapse of the structure or it could indicate numerical difficulties that are not related to the physical behavior. To assist the analyst, default values are given for important parameters. In addition, when divergence is detected, a procedure is followed by which the current stiffness is reformulated and the load step size is reduced. If convergence is still unattainable, structural instability or collapse is likely and should be investigated.

This incremental-iterative solution process is known as the Modified Newton-Raphson Method, and it is described in numerous texts (e.g., Bathe, 1996; Yang and Kuo, 1994).

First-order (linear) analysis, P-Δ analysis, and second-order (nonlinear) analysis all have a place in the design/evaluation of electric transmission structures. While second-order analysis “will always be correct”, it is much more demanding of computer resources than the other two approaches, and obtaining satisfactory results often requires experience and insight. A suggested approach is to begin with a first-order model to define the assemblage of elements, verify loads and support conditions, and gain insight into the behavior of the structure. Design iterations may be performed at this level. Then, evaluate the model using P-Δ assumptions to ensure that the structure is not subject to elastic instability. Finally, second-order analysis may be done to verify that the design performs satisfactorily or to determine collapse behavior.

## 2.2 Element Descriptions

Within the structural analysis module is a library of element types that can be used to generate a structural model. They include:

1. A truss element that can be used to model truss elements, struts, or guys;
2. A frame element, that can be defined as prismatic or tapered, used for poles and cross arms;
3. A general spring/connector element to provide flexible connections between elements;
4. A flexible foundation element; and
5. A cable element for more exact modeling of guys and wires.

In the following sections, the basic assumptions and capabilities of the elements are given. In all cases, a stiffness matrix must be generated, fixed-end forces must be computed for a set of element loads, and, for a given set of displacements, element forces must be produced.

### *2.2.1 Truss element*

The three dimensional truss element is designed to have pinned ends and resist axial force only. It is derived in a local coordinate system, with three degrees of freedom per node as shown in Figure 4.

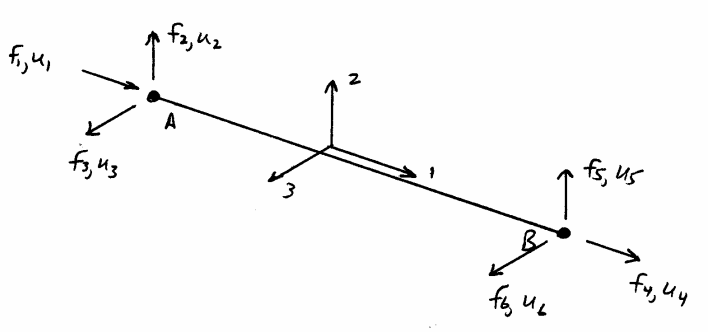


Figure 4. Three dimensional truss element.

The truss element is designed to accept element loads from self-weight, ice, wind, temperature change, and pretension. For second-order analysis, self-weight and pretension are independent of the load multiplier. That is, they are always fully imposed, even without other types of loads.

The tension-only option may be chosen for the truss element so that it can be used to model guys in second-order analysis. As the name implies, if compression is detected in the element, its element force is set to zero and its stiffness is removed. Loads from wind, ice, and self-weight are retained, however.

The geometric stiffness matrix is included for both P-Δ and second-order analysis. Provisions are made for the finite rotation of element forces when second-order analysis is chosen. Details of the element matrices are given in Appendix A.

### *2.2.2 Frame element*

To model poles, cross arms, and any other member subject to bending, a three dimensional frame element is used. It is derived in a local frame of reference, with six degrees of freedom per node as shown in Figure 5.

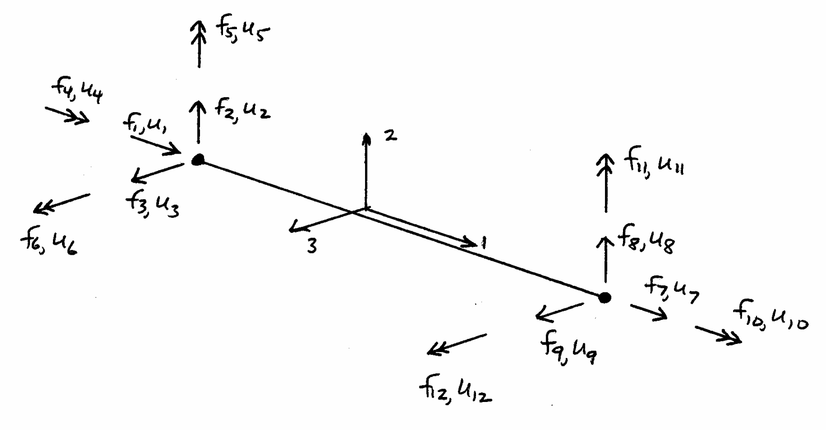


Figure 5. Three dimensional frame element.

The frame element is subject to axial force (degrees of freedom 1 and 7), bending in the local *1-2* plane (degrees of freedom 2, 6, 8, and 12), bending in the local *1-3* plane (degrees of freedom 3, 5, 9, and 11), and torsion (degrees of freedom 4 and 10). The stiffness coefficients are functions of section properties, material properties, and element length. Those for axial stiffness are functions of cross sectional area, *A*, modulus of elasticity, *E*, and element length, *L*. Those for bending in the local *1-2* plane are moment of inertia for bending about the local *3*-axis, $I_3$, *E*, and *L*. Those for bending in the local *1-3* plane are moment of inertia for bending about the local *2*-axis, $I_2$, *E*, and *L*. Those for torsion are the torsion constant, *J*, the shear modulus, *G*, and *L*.

Note that $I_2$ and $I_3$ are the principal moments of inertia and that the line of the element is the axis that runs through the shear center. Also, the element is based on classic beam theory, which includes the assumption that the cross section does not warp. For symmetric, solid, or closed sections, warping is either nonexistent or unimportant and the shear center is at the centroid of the section.

The element may be either prismatic (i.e., a constant cross section along its length) or tapered. The difference is the way in which stiffness matrices and fixed-end forces are generated. Tapered members are assumed to have a linearly varying “radius” that is used to specify the section properties at any point along the element. Types of sections that can be considered include solid round, hollow round, and standard thin-walled hexdecagonal, dodecagonal, octagonal, hexagonal, and square polygons. Although a prismatic member can be designated and modeled as being tapered, the procedure for prismatic elements is simpler and, therefore, less demanding, computationally.

The frame element is designed to accept element load from self-weight, ice, wind, and temperature change. For second-order analysis, self-weight is independent of the load multiplier. In addition, the element may be modified to account for moment releases. Then, static condensation techniques are used to release degrees of freedom 4, 5, 6, 10, 11, and/or 12. Note that torsional degrees of freedom may be released, but if they are released at both ends of the element, a singular stiffness matrix will result.

The effect of the loss of stiffness in bending that results from material cracking or yielding can be considered. Here, a series of moment values for which cracking initiates (designated as "crack points") are defined. When a crack point is detected in an element, the bending stiffness is reduced. There are several assumptions:

1. For prismatic members, with given properties in the principal directions of the cross section, the detection and stiffness reduction in each direction is considered to be independent. For tapered members, assumed to have circular or polygonal cross sections, the vector sum of the moments is considered.
2. Cracking (i.e., stiffness and moment reduction) is based solely on moment. No interaction between moment and axial force or moment and torsion is considered.
3. The reduction in stiffness is included through a reduction in moment of inertia, rather than modulus of elasticity, so that axial stiffness is not affected.
4. A user-defined number of crack points is allowed. The reduction in stiffness is accomplished through a multiplier for each level of cracking. For prismatic elements, the multiplier, which would be less than one, operates on the appropriate moment of inertia, while a separate multiplier operates on the torsion constant to allow a reduction in torsion stiffness. For tapered elements, the multiplier operates on the radius, to reduce the value of moment of inertia and the torsion constant.
5. Unloading and reloading is allowed. The unloading/reloading stiffness multiplier is user-defined. If the member is assumed to behave in a plastic way, the original stiffness would be used. If the stiffness reduction results from material damage/cracking, the user would probably choose a secant stiffness value. In reloading (in either the same direction or opposite direction), the highest previous value of moment experienced by the section becomes the value at which cracking resumes.

In Figure 6, a schematic of the cracking process is shown.

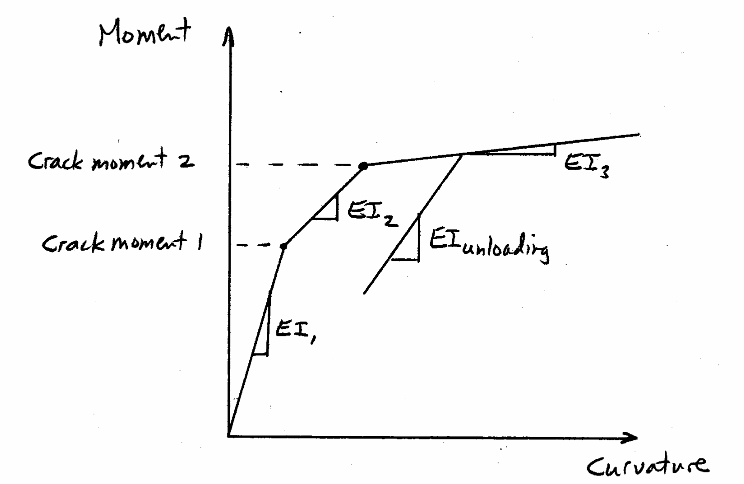


Figure 6. Moment-curvature relationship for bending members with cracking/yielding.

The geometric stiffness matrix is included for both P-Δ and second-order analysis. Provisions are made for finite rotations and deformations within second-order analysis. Details of the element matrices are given in Appendix B.

### *2.2.3 Spring element*

Electric transmission structures often consist of members that are connected with bolts or other flexible connectors. For example, cross arms are typically connected with a bolt that passes through the pole. There are several important considerations:

- The pole and the cross arm members both have finite width and depth. To connect them at a common node, where the node represents the centroid of each member cross section, ignores the moment that results from the offset between them.
- The bolt is not a rigid connection, certainly with respect to its ability to transmit moment. However, while frame elements may be specified to have moment releases in them, that feature is not appropriate for connections between separate members.
- Although they are not rigid in transmitting moment and shear, bolts still contribute some stiffness to the system that may have a significant effect on the distribution of forces.

Figure 7 shows a connection between a cross arm and a pole, along with a schematic showing how the structural model might be constructed. The spring element described here is shown connecting the elements of the cross arm with those of the pole.

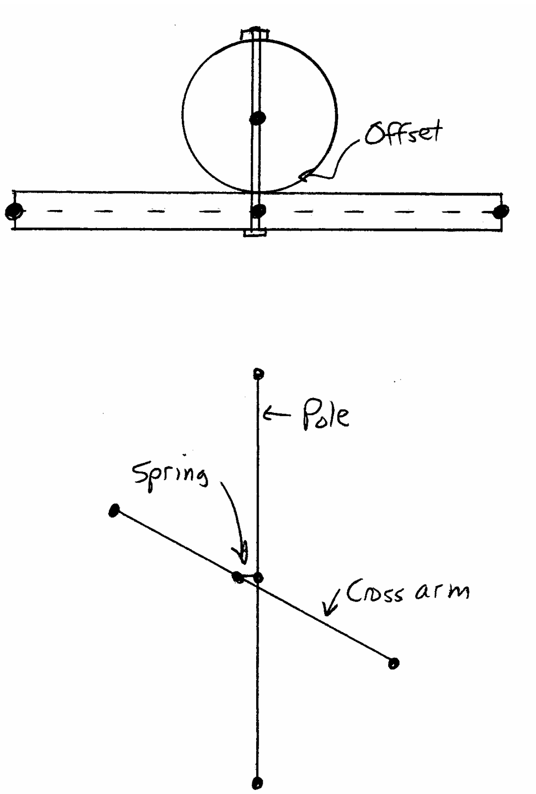


Figure 7. The spring element applied as a connector.

The spring element is derived in essentially the same way as the three dimensional frame element, but with a rotational spring at one end. The layout and degrees of freedom for the element are shown in Figure 8.

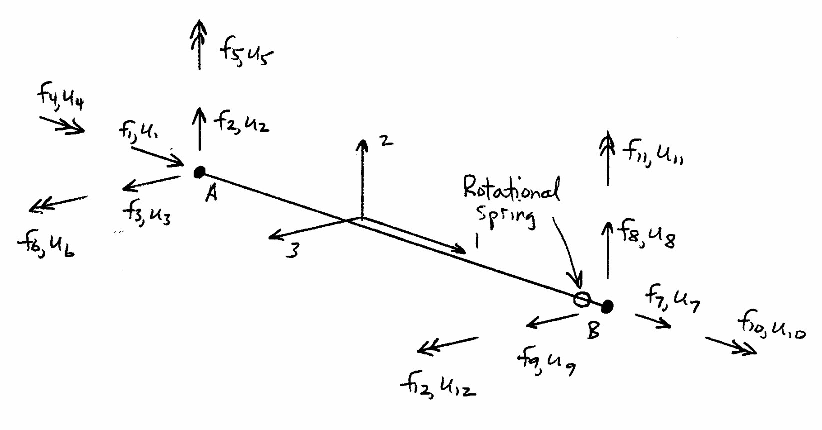


Figure 8. Three dimensional spring element.

As with the frame element, each node has three global translational degrees of freedom and three global rotational degrees of freedom. All of the derivations and capabilities of the frame element are incorporated in the spring element, with the following exceptions:

- Material modulus of elasticity and section moment of inertia are used to define moment and shear stiffness for the element. Axial and torsional stiffness properties are given directly as constants $k_A$ and $k_T$, respectively.
- Because this is a connector element, no capability for member loads was included.
- The flexible rotational spring is allowed at the *b*-end of the element with stiffness values $k_{b1}$ and $k_{b2}$, respectively, for bending in the two principal directions. These values may be set to zero to form a hinge, if desired. For a fixed connection, the values would, theoretically, need to be infinity. In a practical sense, for a continuous element, stiffness should be set to a relatively large value (e.g., 1000 times 4EI/L).

Details of the element matrices are given in Appendix C.

### *2.2.4 Flexible foundation element*

With the flexible foundation element, a linear spring stiffness can be applied as a support for any degree of freedom. This capability is useful for considering the effect of a flexible foundation at the base of a pole or a guy. Translational or rotational springs can be applied, depending on the type of degree of freedom being constrained.

### *2.2.5 Cable element*

In most applications, guys are relatively short, light, and highly tensioned. In those cases, the use of a tension-only truss element to model them is appropriate (ASCE, 1997). In cases of long cables, cables with relatively low pretension, or cables that are significantly loaded with ice, the effect of sag and the nonlinear behavior become significant.

The cable element that has been developed and implemented has the same degrees of freedom and is applied in the same way as the truss element. However, its formulation is based upon the theory of an elastic catenary, as developed by Ahmadi-Kashani and Bell (1988) and presented by Peterson (2002).

The cable element is shown in Figure 9. It is developed in a plane described by the local *1 – 2* axes and, locally, it contributes forces and stiffness only in that plane. As opposed to the other elements described earlier, the local frame of reference does not follow the chord of the element but, rather, is based at one of the ends with the local *1*-axis oriented perpendicular to the distributed applied load, $q_0$.

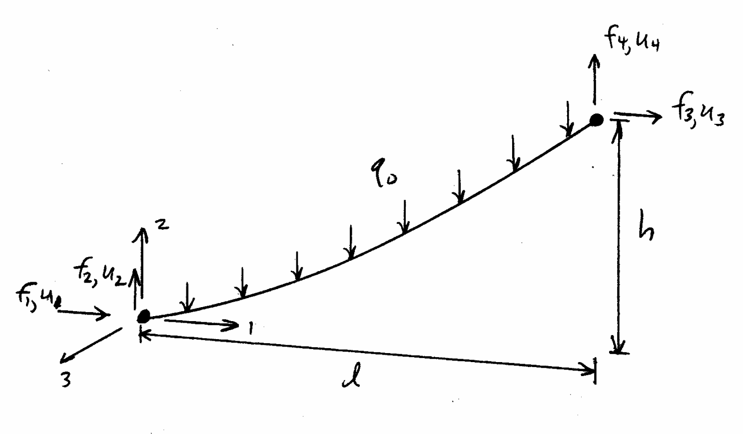


Figure 9.  Cable element.

Because the cable is assumed to be an elastic catenary, the profile is dependent upon the initial, unstretched length and the applied load.  The equilibrium and compatibility equations are inherently nonlinear, which means that an initial pretension cannot be specified directly.  Rather, an iterative, trial and error procedure must be used to determine the unstretched length that results in the desired pretension with the ends of the cable being fixed against movement.  Given that the tension value at the highest end will be greatest, due to the need to support the cable weight, the desired pretension is taken as the average of the tension values at the two ends.

For each cable element, the unstretched length is computed in the initialization phase of the analysis to give the desired pretension, with loading from cable weight alone.  With subsequent displacement of the nodes and additional loading, the cable will sag or tighten appropriately, in accordance with the nonlinear equations for the initial unstretched length.  For any given location of the nodes and any subsequent value of cable load, nodal forces may be computed.

The cable element is designed to accept element load from self-weight, ice, wind, and temperature change.  For second-order analysis, self-weight is independent of the load multiplier.

The stiffness matrix for the cable element is computed by perturbing the nodes and noting the changes in the nodal forces.  When divided by the value of the nodal perturbation, the force values become stiffness coefficients.  By using the nonlinear elastic catenary equations, the resulting stiffness matrix becomes the instantaneous tangent stiffness matrix.  However, because tightening and loosening the cable has different effects, the resulting stiffness matrix is slightly asymmetric.  In order to fit with the other stiffness matrices at the system level, the matrix that is used is the symmetric portion, obtained through decomposition.

For first-order and P-Δ analysis, the stiffness matrix is based on the original location of the nodes and it is not updated.  In that case, the behavior of the element is linearized which may lead to odd results, such as compression in the element.  For second-order analysis, the stiffness matrix is continually updated to reflect the current configuration.

Details of the cable equations and element matrices are given in Appendix D.

## 3.0 References

Ahmadi-Kashani, K. and Bell, A. J. (1988). "The Analysis of Cable Subject to Uniformly Distributed Loads," *Engineering Structures,* Vol. 10, No. 3, pp. 174-184.

ASCE (1997). "Design of Guyed Electrical Transmission Structures," ASCE Manuals and Reports on Engineering Practice No. 91, American Society of Civil Engineers, New York, NY.

Bathe, K. J. (1996). *Finite Element Procedures*, Prentice-Hall, Inc., Englewood Cliffs, NJ.

Cook, R. D., Malkus, D. S., Plesha, M. E., and Witt, R. J. (2002). *Concepts and Applications of Finite Element Analysis*, 4th edition, John Wiley & Sons, Inc.

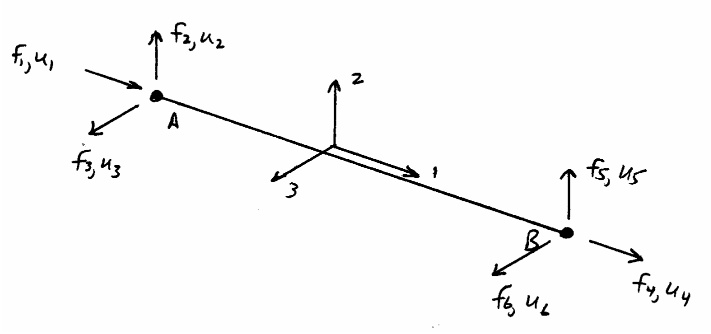
McGuire, W., Gallagher, R. H., and Ziemian, R. D. (2000). *Matrix Structural Analysis,* 2nd Edition, John Wiley & Sons, Inc., New York, NY.

Peterson, S. T. (2002). "Experimental Response and Analysis of the Evergreen Point Floating Bridge," dissertation submitted in partial fulfillment of the requirements for the degree of Doctor of Philosophy in Civil Engineering, Washington State University, Pullman, WA.

USDA (1992). "Design Manual for High Voltage Transmission Lines," United States Department of Agriculture, Rural Electrification Administration, REA Bulletin 1724E-200, Washington, DC.

Yang, Y. B. and Kuo, S. R. (1994). *Theory and Analysis of Nonlinear Framed Structures,* Prentice-Hall, Simon & Schuster (Asia) Pte Ltd., Singapore.

# Appendix A
## Matrices for the Truss Element



### Stiffness matrix

The first-order stiffness matrix for a truss in the local frame of reference is well known:

$$\underline{\underline{k}}' = \frac{AE}{L}\begin{bmatrix} 1 & 0 & 0 & -1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ -1 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix}$$

where *A* is the cross sectional area, *E* is the modulus of elasticity, *L* is the element length, and the prime indicates that it is defined in the local frame of reference. To transform the stiffness matrix to the global frame of reference, the direction cosines for the unit vector defined by the element, $V_1$, are given as *l*, *m*, and *n*. Then, the first-order stiffness matrix in the global frame of reference is:

$$\underline{\underline{k}} = \frac{AE}{L}\begin{bmatrix} l^2 & ml & nl & -l^2 & -ml & -nl \\ ml & m^2 & mn & -ml & -m^2 & -mn \\ nl & mn & n^2 & -nl & -mn & -n^2 \\ -l^2 & -ml & -nl & l^2 & ml & nl \\ -ml & -m^2 & -mn & ml & m^2 & mn \\ -nl & -mn & -n^2 & nl & mn & n^2 \end{bmatrix}$$

The geometric stiffness matrix, required for P-Δ and second-order analysis, is also well known and given in the local frame of reference as:

$$\underline{\underline{k}}'_g = \frac{P}{L}\begin{bmatrix} 1 & 0 & 0 & -1 & 0 & 0 \\ 0 & 1 & 0 & 0 & -1 & 0 \\ 0 & 0 & 1 & 0 & 0 & -1 \\ -1 & 0 & 0 & 1 & 0 & 0 \\ 0 & -1 & 0 & 0 & 1 & 0 \\ 0 & 0 & -1 & 0 & 0 & 1 \end{bmatrix}$$

where $P$ is taken as the average axial force, defined as

$$P = \frac{1}{2}\left(f_4 - f_1\right)$$

Note that $P$ will be positive for a tensile force and negative for a compressive force. The local geometric stiffness matrix must be transformed to the global frame of reference. This is a bit more complicated than the transformation for the first-order stiffness matrix because nonzero components now exist for local $v$ and $w$ degrees of freedom. The orientation of the local *2* and *3*-axes must be defined. The following procedure was used.

First, an arbitrary vector, $V_{12}$, is defined, different from the vector that forms the element, that is declared as being in the local *1 – 2* plane. The convention chosen was that, unless the element is oriented in the global $y$-direction, $V_{12}$ has global components (0,1,0). Otherwise, $V_{12}$ has global components (1,0,0). Then,

$$\underline{V}_3 = \underline{V}_1 \times \underline{V}_{12} = \begin{Bmatrix} l_3 \\ m_3 \\ n_3 \end{Bmatrix}, \text{ and}$$

$$\underline{V}_2 = \underline{V}_3 \times \underline{V}_1 = \begin{Bmatrix} l_2 \\ m_2 \\ n_2 \end{Bmatrix}$$

The transformation matrix may now be formed as

$$\underline{\underline{T}} = \begin{bmatrix} l & m & n & 0 & 0 & 0 \\ l_2 & m_2 & n_2 & 0 & 0 & 0 \\ l_3 & m_3 & n_3 & 0 & 0 & 0 \\ 0 & 0 & 0 & l & m & n \\ 0 & 0 & 0 & l_2 & m_2 & n_2 \\ 0 & 0 & 0 & l_3 & m_3 & n_3 \end{bmatrix},$$

and the geometric stiffness matrix in the global frame of reference is

$$\underline{\underline{k}}_g = \underline{\underline{T}}^T \underline{\underline{k'}}_g \underline{\underline{T}}$$

**Fixed-end forces**

Fixed-end forces from element loads in the truss element result from element weight, distributed ice load, wind, temperature change, and pretension.

*Element weight*

Components of material weight density in the global frame of reference, $(g_x, g_y, g_z)$, are provided. Then, the vector of fixed-end forces is:

$$\underline{fef}_1 = \frac{AL}{2}\begin{Bmatrix} -g_x \\ -g_y \\ -g_z \\ -g_x \\ -g_y \\ -g_z \end{Bmatrix}$$

*Ice load*

Components of ice load in the global frame of reference, $(w_x, w_y, w_z)$, are provided, with units of force per length. Then, the vector of fixed-end forces is:

$$\underline{fef}_2 = \frac{L}{2}\begin{Bmatrix} -w_x \\ -w_y \\ -w_z \\ -w_x \\ -w_y \\ -w_z \end{Bmatrix}$$

*Effect of temperature change*

The coefficient of thermal expansion, $a$, and the change in temperature, *DT*, are provided. Then, the vector of fixed-end forces is:

$$\underline{fef}_3 = AEa\Delta T \begin{Bmatrix} l \\ m \\ n \\ -l \\ -m \\ -n \end{Bmatrix}$$

*Pretension*

Intended for use mainly when modeling guys, the pretension, *T*, is provided. Then, the vector of fixed-end forces is:

$$\underline{fef}_4 = T \begin{Bmatrix} -l \\ -m \\ -n \\ l \\ m \\ n \end{Bmatrix}$$

The total fixed-end force vector for the element is:

$$\underline{fef}_{member} = \underline{fef}_1 + \underline{fef}_4 + l\left(\underline{fef}_2 + \underline{fef}_3\right)$$

where $l$ is the load multiplier.

### Element Forces

Once the displacements for the nodes have been determined, element forces may be computed. The manner in which they are computed varies depending on which type of analysis is being performed.

*First-order analysis*

Because the structural properties are assumed to remain unchanged in first-order analysis, the computation of element force is quite straightforward. First, the node displacements, with components in the global frame of reference, are gathered.

$$\underline{u}_{member} = \begin{Bmatrix} u_A \\ v_A \\ w_A \\ u_B \\ v_B \\ w_B \end{Bmatrix}$$

where *A* and *B* refer to the ends of the element, shown in Figure 4. Then, the global element force components are:

$$\underline{f}_{member} = \underline{\underline{k}}\,\underline{u}_{member} + \underline{fef}_{member}$$

Local axial force components are:

$$\begin{aligned} f_1' &= lf_1 + mf_2 + nf_3 \\ f_4' &= lf_4 + mf_5 + nf_6 \end{aligned}$$

*P-D analysis*

The procedure for computing element forces is identical to that for first-order analysis except that $\underline{\underline{k}}_t$ is used instead of $\underline{\underline{k}}$.

*Second-order analysis*

Here, provision must be made for the rotation of the element. First, knowing the displacement of the nodes, their coordinates are updated. With new node coordinates, the direction cosines that orient the element are updated. Using the resulting transformation matrix, *T*, the global node displacement components are transformed to components in the current local frame of reference.

$$\underline{u}'_{member} = \underline{\underline{T}}\,\underline{u}_{member}$$

The incremental axial force is computed using the concept of Almansi strain (i.e., the element has rotated from the previous orientation to the current one, which is the frame of reference.). Note that, although finite rotations are considered, strains are assumed to be small. Thus, the original cross sectional area and the original modulus of elasticity are used. Define:

$$\begin{aligned} \Delta u &= u_B' - u_A' \\ \Delta v &= v_B' - v_A' \\ \Delta w &= w_B' - w_A' \end{aligned}$$

Then, the axial force that results from this displacement is

$$\Delta f' = EA\left(\frac{\Delta u}{L} - \frac{1}{2}\left[\left(\frac{\Delta u}{L}\right)^2 + \left(\frac{\Delta v}{L}\right)^2 + \left(\frac{\Delta w}{L}\right)^2\right]\right)$$

Local element forces are updated. Note that the forces being considered here are based on node displacements only, separate from fixed-end forces. Therefore, the accumulated elastic force for each element, designated as $\underline{f}'^{elastic}_{old}$, is stored as a history variable.

$$\begin{aligned}
\left(f'^{elastic}_{new}\right)_1 &= \left(f'^{elastic}_{old}\right)_1 - \Delta f' \\
\left(f'^{elastic}_{new}\right)_4 &= \left(f'^{elastic}_{old}\right)_4 + \Delta f'
\end{aligned}$$

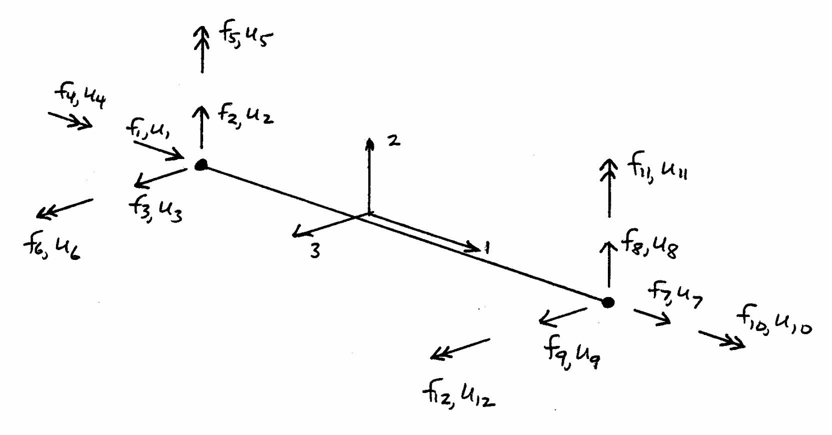
Updated local elastic element forces are transformed into global components, using the direction cosines for the current configuration and fixed-end forces in the current configuration are added to obtain the current complete global element force components.

$$\underline{f}_{new} = \left(f'^{elastic}_{new}\right)_1 \begin{Bmatrix} l \\ m \\ n \\ 0 \\ 0 \\ 0 \end{Bmatrix} + \left(f'^{elastic}_{new}\right)_4 \begin{Bmatrix} 0 \\ 0 \\ 0 \\ l \\ m \\ n \end{Bmatrix} + \underline{fef}_{member}$$

In the updated local frame of reference, the axial element forces are:

$$\begin{aligned}
\left(f'_{new}\right)_1 &= l\left(f_{new}\right)_1 + m\left(f_{new}\right)_2 + n\left(f_{new}\right)_3 \\
\left(f'_{new}\right)_4 &= l\left(f_{new}\right)_4 + m\left(f_{new}\right)_5 + n\left(f_{new}\right)_6
\end{aligned}$$

# Appendix B
# Matrices for the Frame Element



## Stiffness matrices

### *Prismatic element*

The first-order stiffness matrix for a prismatic frame element in the local frame of reference is well known:

$$\underline{\underline{k}}' = \begin{bmatrix}
\frac{AE}{L} & 0 & 0 & 0 & 0 & 0 & -\frac{AE}{L} & 0 & 0 & 0 & 0 & 0 \\
0 & \frac{12EI_3}{L^3} & 0 & 0 & 0 & \frac{6EI_3}{L^2} & 0 & -\frac{12EI_3}{L^3} & 0 & 0 & 0 & \frac{6EI_3}{L^2} \\
0 & 0 & \frac{12EI_2}{L^3} & 0 & -\frac{6EI_2}{L^2} & 0 & 0 & 0 & -\frac{12EI_2}{L^3} & 0 & -\frac{6EI_2}{L^2} & 0 \\
0 & 0 & 0 & \frac{JG}{L} & 0 & 0 & 0 & 0 & 0 & -\frac{JG}{L} & 0 & 0 \\
0 & 0 & -\frac{6EI_2}{L^2} & 0 & \frac{4EI_2}{L} & 0 & 0 & 0 & \frac{6EI_2}{L^2} & 0 & \frac{2EI_2}{L} & 0 \\
0 & \frac{6EI_3}{L^2} & 0 & 0 & 0 & \frac{4EI_3}{L} & 0 & -\frac{6EI_3}{L^2} & 0 & 0 & 0 & \frac{2EI_3}{L} \\
-\frac{AE}{L} & 0 & 0 & 0 & 0 & 0 & \frac{AE}{L} & 0 & 0 & 0 & 0 & 0 \\
0 & -\frac{12EI_3}{L^3} & 0 & 0 & 0 & -\frac{6EI_3}{L^2} & 0 & \frac{12EI_3}{L^3} & 0 & 0 & 0 & -\frac{6EI_3}{L^2} \\
0 & 0 & -\frac{12EI_2}{L^3} & 0 & \frac{6EI_2}{L^2} & 0 & 0 & 0 & \frac{12EI_2}{L^3} & 0 & \frac{6EI_2}{L^2} & 0 \\
0 & 0 & 0 & -\frac{JG}{L} & 0 & 0 & 0 & 0 & 0 & \frac{JG}{L} & 0 & 0 \\
0 & 0 & -\frac{6EI_2}{L^2} & 0 & \frac{2EI_2}{L} & 0 & 0 & 0 & \frac{6EI_2}{L^2} & 0 & \frac{4EI_2}{L} & 0 \\
0 & \frac{6EI_3}{L^2} & 0 & 0 & 0 & \frac{2EI_3}{L} & 0 & -\frac{6EI_3}{L^2} & 0 & 0 & 0 & \frac{4EI_3}{L}
\end{bmatrix}$$

where $A$ is cross sectional area, $E$ is modulus of elasticity, $L$ is member length, $I_2$ is moment of inertia for bending in the local *1-3* plane, $I_3$ is moment of inertia for bending in the local *1-2* plane, $J$ is the torsion constant, and $G$ is the shear modulus.

The element stiffness matrix must be transformed into the global frame of reference. The transformation matrix that is used contains the direction cosines of the three local coordinate axes. The direction cosines of the local *1*-axis are unique and easily obtained from the unit vector, $\underline{V}_1 = \begin{Bmatrix} l \\ m \\ n \end{Bmatrix}$, that is in the direction of the element. However, three-dimensional beams have the additional complication of having to orient the principal axes of bending. Two methods are allowed to orient the local *1-2* plane.

The first, and most straightforward way, is to specify a vector, $V_{12}$, that is parallel to one of the global axes and is used to define the direction of the *1-2* plane. That is, if the global *x*-axis is chosen, $\underline{V}_{12} = \begin{Bmatrix} 1 \\ 0 \\ 0 \end{Bmatrix}$. Similarly, if the global *y*-axis is chosen, $\underline{V}_{12} = \begin{Bmatrix} 0 \\ 1 \\ 0 \end{Bmatrix}$, and if the global *z*-axis is chosen, $\underline{V}_{12} = \begin{Bmatrix} 0 \\ 0 \\ 1 \end{Bmatrix}$. Then, the vector that is the local *3*-axis is defined as:

$$\underline{V}_3 = \underline{V}_1 \times \underline{V}_{12} = \begin{Bmatrix} l_3 \\ m_3 \\ n_3 \end{Bmatrix}$$

The vector that is the local *2*-axis may now be defined:

$$\underline{V}_2 = \underline{V}_3 \times \underline{V}_1 = \begin{Bmatrix} l_2 \\ m_2 \\ n_2 \end{Bmatrix}$$

The other way in which the principal axes may be oriented is to specify an auxiliary node that, with the two nodes of the element, defines the local *1-2* plane. Here, the unit vector, $V_{12}$, is in the direction from the node at the *A*-end of the element to the auxiliary node. Then, $V_3$ and $V_2$ are defined as before.

Given the direction cosines, the transformation matrix, $\underline{\underline{T}}$, is defined as:

$$\underline{\underline{T}} = \begin{bmatrix} l & m & n & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ l_2 & m_2 & n_2 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ l_3 & m_3 & n_3 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & l & m & n & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & l_2 & m_2 & n_2 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & l_3 & m_3 & n_3 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & l & m & n & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & l_2 & m_2 & n_2 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & l_3 & m_3 & n_3 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & l & m & n \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & l_2 & m_2 & n_2 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & l_3 & m_3 & n_3 \end{bmatrix}$$

The first-order stiffness matrix in the global reference frame is:

$$\underline{\underline{k}} = \underline{\underline{T}}^T \underline{\underline{k'}}\,\underline{\underline{T}}$$

*Tapered element*

The basis for the development of the tapered beam element is that of the Finite Element Method, in which the stiffness matrix of any type of element is derived from strain energy. For example, stiffness for a bending element in a plane may be given as:

$$\underline{\underline{k'_b}} = \int_0^L \underline{\underline{B}}^T EI\,\underline{\underline{B}}dx$$

where $\underline{\underline{B}}$ is based on an assumed set of shape functions and provides curvature at any given point, *x*, as a function of nodal displacements. Similarly, stiffness for an axial element may be given as:

$$\underline{\underline{k'}}_a = \int_0^L \underline{\underline{B}}^T EA\underline{\underline{B}}dx$$

where, in this case, $\underline{\underline{B}}$ provides axial stretch along the length as a function of nodal displacements. For a torsion member,

$$\underline{\underline{k'}}_T = \int_0^L \underline{\underline{B}}^T GJ\,\underline{\underline{B}}dx$$

where $\underline{\underline{B}}$ provides twist along the length as a function of nodal displacements.

This approach was chosen because, as opposed to prismatic beams in which *I*, *A*, and *J* are constant, the variation of the section properties can be naturally included as part of the integrals. Following the ideas of the Finite Element Method, stiffness coefficients are computed by defining shape functions, developing the appropriate $\underline{\underline{B}}$ matrix, and integrating numerically. Each contributor to the stiffness matrix is considered independently, as follows:

Bending

Here, we assume a natural coordinate, *x*, which varies from 0 to 1 along the length of the member (i.e., $x = \frac{x}{L}$). Then, the shape functions for bending within the 1-2 plane are given as:

$$\underline{\underline{N}} = \left[1 - 3x^2 + 2x^3 \quad L\left(x - 2x^2 + x^3\right) \quad 3x^2 - 2x^3 \quad L\left(-x^2 + x^3\right)\right]$$

The curvature-displacement relationship is obtained by taking the second derivative with respect to *x*. It is given as:

$$\underline{\underline{B}} = \left[\frac{6}{L^2}(-1+2x) \quad \frac{2}{L}(-2+3x) \quad \frac{6}{L^2}(1-2x) \quad \frac{2}{L}(-1+3x)\right]$$

Then, on the basis of strain energy, the stiffness matrix is defined as:

$$\underline{\underline{k_b'}} = \int_0^L \underline{\underline{B}}^T EI \underline{\underline{B}} dx$$

Here, for a tapered member, *I* is a function of *x*. Then, switching variables from *x* to *x* and first considering bending in the *1-2* plane,

$$\underline{\underline{k_{b12}'}} = \int_0^1 EI_3(x) \begin{Bmatrix} \frac{6}{L^2}(-1+2x) \\ \frac{2}{L}(-2+3x) \\ \frac{6}{L^2}(1-2x) \\ \frac{2}{L}(-1+3x) \end{Bmatrix} \left[\frac{6}{L^2}(-1+2x) \quad \frac{2}{L}(-2+3x) \quad \frac{6}{L^2}(1-2x) \quad \frac{2}{L}(-1+3x)\right] L dx$$

Expand, noting symmetry:

$$\underline{\underline{k'_{b12}}} = \int_0^1 EI_3(x)\begin{bmatrix} \frac{36}{L^3}(-1+2x)^2 & \frac{12}{L^2}(-1+2x)(-2+3x) & \frac{36}{L^3}(-1+2x)(1-2x) & \frac{12}{L^2}(-1+2x)(-1+3x) \\ & \frac{4}{L}(-2+3x)^2 & \frac{12}{L^2}(-2+3x)(1-2x) & \frac{4}{L}(-2+3x)(-1+3x) \\ & & \frac{36}{L^3}(1-2x)^2 & \frac{12}{L^2}(1-2x)(-1+3x) \\ & & & \frac{4}{L}(-1+3x)^2 \end{bmatrix} dx$$

Integration is performed numerically, using $n$ integration points, $x_i$, and weight factors, $W_i$. Then,

$$\underline{\underline{k'_{b12}}} = \sum_{i=1}^{n} EI_3(x_i)\begin{bmatrix} \frac{36}{L^3}\left(1-4x_i+4x_i^2\right) & \frac{12}{L^2}\left(2-7x_i+6x_i^2\right) & \frac{36}{L^3}\left(-1+4x_i-4x_i^2\right) & \frac{12}{L^2}\left(1-5x_i+6x_i^2\right) \\ & \frac{4}{L}\left(4-12x_i+9x_i^2\right) & \frac{12}{L^2}\left(-2+7x_i-6x_i^2\right) & \frac{4}{L}\left(2-9x_i+9x_i^2\right) \\ & & \frac{36}{L^3}\left(1-4x_i+4x_i^2\right) & \frac{12}{L^2}\left(-1+5x_i-6x_i^2\right) \\ & & & \frac{4}{L}\left(1-6x_i+9x_i^2\right) \end{bmatrix} W_i$$

This matrix pertains to degrees of freedom 2, 6, 8, and 12. Similarly, for bending in the *1-3* plane, the stiffness matrix is:

$$\underline{\underline{k'_{b13}}} = \sum_{i=1}^{n} EI_2(x_i)\left[\begin{matrix} \frac{36}{L^3}\left(1-4x_i+4x_i^2\right) & -\frac{12}{L^2}\left(2-7x_i+6x_i^2\right) & \frac{36}{L^3}\left(-1+4x_i-4x_i^2\right) & -\frac{12}{L^2}\left(1-5x_i+6x_i^2\right) \\ & \frac{4}{L}\left(4-12x_i+9x_i^2\right) & -\frac{12}{L^2}\left(-2+7x_i-6x_i^2\right) & \frac{4}{L}\left(2-9x_i+9x_i^2\right) \\ & & \frac{36}{L^3}\left(1-4x_i+4x_i^2\right) & -\frac{12}{L^2}\left(-1+5x_i-6x_i^2\right. \\ & & & \frac{4}{L}\left(1-6x_i+9x_i^2\right) \end{matrix}\right.$$

This matrix pertains to degrees of freedom 3, 5, 9, and 11.

<u>Axial stiffness</u>

The stiffness coefficients for axial force are derived similarly. Shape functions are:

$$\underline{\underline{N}} = \begin{bmatrix} 1-x & x \end{bmatrix}$$

The strain-displacement matrix is:

$$\underline{\underline{B}} = \begin{bmatrix} -\frac{1}{L} & \frac{1}{L} \end{bmatrix}$$

The stiffness matrix becomes:

$$\underline{\underline{k'_a}} = \int_0^1 \underline{\underline{B}}^T AE\underline{\underline{B}}L dx$$

where $A$ is a function of $x$. Plugging in and multiplying:

$$\underline{\underline{k'_a}} = \int_0^1 A(x)E \begin{bmatrix} \frac{1}{L} & -\frac{1}{L} \\ -\frac{1}{L} & \frac{1}{L} \end{bmatrix} dx$$

Integrating numerically:

$$\underline{\underline{k'_a}} = \sum_{i=1}^{n} \begin{bmatrix} \frac{A(x_i)E}{L} & -\frac{A(x_i)E}{L} \\ -\frac{A(x_i)E}{L} & \frac{A(x_i)E}{L} \end{bmatrix} W_i$$

This matrix pertains to degrees of freedom 1 and 7.

Torsion stiffness

For torsion, the exact same approach is used, resulting in:

$$\underline{\underline{k'_T}} = \sum_{i=1}^{n} \begin{bmatrix} \frac{J(x_i)G}{L} & -\frac{J(x_i)G}{L} \\ -\frac{J(x_i)G}{L} & \frac{J(x_i)G}{L} \end{bmatrix} W_i$$

This matrix pertains to degrees of freedom 4 and 10.

Numerical integration

The number of integration points that are needed for the various contributors to the stiffness matrix must be determined. For the type of quadrature being used, a polynomial of degree $2n-1$ is integrated exactly, where $n$ is the number of points. If a circular cross section is assumed, and radius can vary linearly along the length of the member, then $I$

varies as $x^4$ (remembering that $I = \frac{pr^4}{4}$). So, a polynomial of the 6th degree is being integrated, requiring 4 points. For bending, then,

$$\text{Integration points, } x = \begin{Bmatrix} 0.069431845 \\ 0.330009479 \\ 0.669990522 \\ 0.930568156 \end{Bmatrix}, \text{ weight factors, } W = \begin{Bmatrix} 0.173927423 \\ 0.326072578 \\ 0.326072578 \\ 0.173927423 \end{Bmatrix}.$$

For the axial stiffness coefficients, the cross sectional area varies quadratically, requiring 2 points.

$$\text{Integration points, } x = \begin{Bmatrix} 0.211324866 \\ 0.788675135 \end{Bmatrix}, \text{ weight factors, } W = \begin{Bmatrix} 0.5 \\ 0.5 \end{Bmatrix}.$$

For the torsional stiffness coefficients, the polar moment of inertia varies as a polynomial of degree 4, which requires 3 points.

$$\text{Integration points, } x = \begin{Bmatrix} 0.112701666 \\ 0.5 \\ 0.887298335 \end{Bmatrix}, \text{ weight factors, } W = \begin{Bmatrix} 0.277777778 \\ 0.444444444 \\ 0.277777778 \end{Bmatrix}.$$

The stiffness contributions from bending, axial force, and torsion are combined to form the total element stiffness matrix in the local frame of reference. The orientation of the local axes and the transformation matrix are the same as those for the prismatic element.

*Geometric stiffness matrix*

The geometric stiffness matrix for the three dimensional frame element is derived from the principle of virtual work, as outlined in the text by Yang and Kuo (1994), pages 354 – 359. First, the following constants are defined, which are written in terms of the existing element forces, cross sectional area, *A*, member length, *L*, moments of inertia, $I_2$ and $I_3$, and torsion constant, *J*.

$$f_{avg} = \frac{1}{2}\left(f_7 - f_1\right),$$

$$a = \frac{f_{avg}}{L},\ b = 1.2a + \frac{12 f_{avg} I_3}{AL^3},\ c = 1.2a + \frac{12 f_{avg} I_2}{AL^3},\ d = \frac{f_5}{L},\ e = \frac{f_6}{L},$$

$$f = \frac{f_{avg} J}{AL},\ g = \frac{f_{10}}{L},\ h = \frac{f_{avg}}{10} + \frac{6 f_{avg} I_2}{AL^2},\ i = \frac{f_6 + f_{12}}{6},\ j = \frac{2 f_{avg} L}{15} + \frac{4 f_{avg} I_2}{AL},$$

$$k = \frac{f_{avg}}{10} + \frac{6 f_{avg} I_3}{AL^2},\ l = -\frac{f_5 + f_{11}}{6},\ m = \frac{2 f_{avg} L}{15} + \frac{4 f_{avg} I_3}{AL},\ n = \frac{f_{11}}{L},\ o = \frac{f_{12}}{L},$$

$$p = -\frac{f_{avg}L}{30} + \frac{2 f_{avg} I_2}{AL},\ q = -\frac{f_{10}}{2},\ r = -\frac{f_{avg}L}{30} + \frac{2 f_{avg} I_3}{AL}$$

The geometric stiffness matrix is defined as:

$$\underline{\underline{k}}_g' = \begin{bmatrix} a & 0 & 0 & 0 & -d & -e & -a & 0 & 0 & 0 & -n & -o \\ 0 & b & 0 & d & g & k & 0 & -b & 0 & n & -g & k \\ 0 & 0 & c & e & -h & g & 0 & 0 & -c & o & -h & -g \\ 0 & d & e & f & i & l & 0 & -d & -e & -f & -i & -l \\ -d & g & -h & i & j & 0 & d & -g & h & -i & p & -q \\ -e & k & g & l & 0 & m & e & -k & -g & -l & q & r \\ -a & 0 & 0 & 0 & d & e & a & 0 & 0 & 0 & n & o \\ 0 & -b & 0 & -d & -g & -k & 0 & b & 0 & -n & g & -k \\ 0 & 0 & -c & -e & h & -g & 0 & 0 & c & -o & h & g \\ 0 & n & o & -f & -i & -l & 0 & -n & -o & f & i & l \\ -n & -g & -h & -i & p & q & n & g & h & i & j & 0 \\ -o & k & -g & -l & -q & r & o & -k & g & l & 0 & m \end{bmatrix}$$

In addition, Yang and Kuo (1994, pp. 367 – 371) present modifications that result from the effect of joint rotation on torque and vice versa. This supplemental stiffness matrix is called the *joint moment matrix,* $\underline{\underline{k}}_j$ .

$$\underline{\underline{k}}'_j = \begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & -\frac{f_6}{2} & \frac{f_5}{2} & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & -\frac{f_6}{2} & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{f_5}{2} & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & -\frac{f_{12}}{2} & \frac{f_{11}}{2} \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & -\frac{f_{12}}{2} & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & \frac{f_{11}}{2} & 0 & 0 \end{bmatrix}$$

Regarding the geometric stiffness matrix for the tapered element, an examination of the contents reveals that most of the terms are independent of the cross sectional properties.

This is reasonable because we know that geometric effects come from the rotation of forces and moments. However, constants *b, c, h, j, k, m, p,* and *r* have moment of inertia and cross sectional area in them. These result from the following terms in the virtual work expression, given on p. 350 of Yang and Kuo (1994):

$$\frac{1}{2}\int_0^L {}^1F_x \frac{I_2}{A} dw,_{xx}{}^2\, dx \text{ and } \frac{1}{2}\int_0^L {}^1F_x \frac{I_3}{A} dv,_{xx}{}^2\, dx\,.$$

Using the same shape functions as were used for the first-order stiffness matrix of the tapered element, these expressions become:

$\int_0^L \underline{\underline{B}}^T \left( {}^1F_x \frac{I_2}{A} \right) \underline{\underline{B}} dx$ for displacement in the *w* direction, and $\int_0^L \underline{\underline{B}}^T \left( {}^1F_x \frac{I_3}{A} \right) \underline{\underline{B}} dx$ for displacement in the *v* direction. Interestingly, these terms have the same form as those of the bending stiffness matrices, except that *EI* is replaced by ${}^1F_x \frac{I}{A}$. Thus, they can be formed in exactly the same way and added to the remainder of the geometric stiffness matrix. The following matrix pertains to degrees of freedom 2, 6, 8, and 12:

$$\underline{\underline{k'}}_{g1} = \sum_{i=1}^{n} f_{avg} \frac{I_3(x_i)}{A(x_i)} \begin{bmatrix} \frac{36}{L^3}\left(1-4x_i+4x_i{}^2\right) & \frac{12}{L^2}\left(2-7x_i+6x_i{}^2\right) & \frac{36}{L^3}\left(-1+4x_i-4x_i{}^2\right) & \frac{12}{L^2}\left(1-5x_i+6x_i{}^2\right) \\ & \frac{4}{L}\left(4-12x_i+9x_i{}^2\right) & \frac{12}{L^2}\left(-2+7x_i-6x_i{}^2\right) & \frac{4}{L}\left(2-9x_i+9x_i{}^2\right) \\ & & \frac{36}{L^3}\left(1-4x_i+4x_i{}^2\right) & \frac{12}{L^2}\left(-1+5x_i-6x_i{}^2\right) \\ & & & \frac{4}{L}\left(1-6x_i+9x_i{}^2\right) \end{bmatrix}$$

The following matrix pertains to degrees of freedom 3, 5, 9, and 11:

$$\underline{\underline{k'}}_{g2} = \sum_{i=1}^{n} f_{avg} \frac{I_2(x_i)}{A(x_i)} \left[ \begin{matrix} \frac{36}{L^3}\left(1-4x_i+4x_i{}^2\right) & -\frac{12}{L^2}\left(2-7x_i+6x_i{}^2\right) & \frac{36}{L^3}\left(-1+4x_i-4x_i{}^2\right) & -\frac{12}{L^2}\left(1-5x_i+6\right. \\ & \frac{4}{L}\left(4-12x_i+9x_i{}^2\right) & -\frac{12}{L^2}\left(-2+7x_i-6x_i{}^2\right) & \frac{4}{L}\left(2-9x_i+9x\right. \\ & & \frac{36}{L^3}\left(1-4x_i+4x_i{}^2\right) & -\frac{12}{L^2}\left(-1+5x_i-\right. \\ & & & \frac{4}{L}\left(1-6x_i+9x\right. \end{matrix} \right.$$

*Moment releases*

The insertion of a hinge at one or both ends of the element is often required to accurately model structural behavior. For the three dimensional frame element, whether prismatic or tapered, well known techniques are used (e.g., Cook, et al., 2002). That is, static condensation is employed to remove the stiffness for specific rotational degrees of freedom.

*Complete stiffness matrix*

The complete tangent stiffness matrix for the frame element in the local frame of reference is:

$$\underline{\underline{k}}'_t = \underline{\underline{k}}' + \underline{\underline{k}}'_g + \underline{\underline{k}}'_j$$

In global components,

$$\underline{\underline{k}}_t = \underline{\underline{T}}^T \underline{\underline{k}}'_t \underline{\underline{T}}$$

### Fixed-end Forces

Fixed-end forces from element loads on the three dimensional frame element result from element weight, distributed ice load, wind, and temperature change.

*Prismatic element*

The fixed-end forces for prismatic elements are well known and given as simple formulas in texts. The only consideration is that the formulas are derived for load components given in the local frame of reference.

Member weight

Global components of material weight density, $(g_x, g_y, g_z)$, are provided. Global components of distributed force are obtained by multiplying by cross sectional area. For a differential length of the element, its local force components are determined by transforming into local components:

$$\begin{Bmatrix} w'_{x2} \\ w'_{y2} \\ w'_{z2} \end{Bmatrix} = A \cdot \begin{bmatrix} l & m & n \\ l_2 & m_2 & n_2 \\ l_3 & m_3 & n_3 \end{bmatrix} \begin{Bmatrix} g_x \\ g_y \\ g_z \end{Bmatrix}$$

Then, the fixed-end force vector in the local frame of reference is:

$$\underline{fef'}_1 = \begin{Bmatrix} -\dfrac{w'_{x2}L}{2} \\ -\dfrac{w'_{y2}L}{2} \\ -\dfrac{w'_{z2}L}{2} \\ 0 \\ \dfrac{w'_{z2}L^2}{12} \\ -\dfrac{w'_{y2}L^2}{12} \\ -\dfrac{w'_{x2}L}{2} \\ -\dfrac{w'_{y2}L}{2} \\ -\dfrac{w'_{z2}L}{2} \\ 0 \\ -\dfrac{w'_{z2}L^2}{12} \\ \dfrac{w'_{y2}L^2}{12} \end{Bmatrix}$$

Ice load

Global components of ice load in the global frame of reference, $(w_x, w_y, w_z)$, are provided, in units of force per length. The procedure to determine fixed-end forces is the same as that used for element weight. The local force components are:

$$\begin{Bmatrix} w'_x \\ w'_y \\ w'_z \end{Bmatrix} = \begin{bmatrix} l & m & n \\ l_2 & m_2 & n_2 \\ l_3 & m_3 & n_3 \end{bmatrix} \begin{Bmatrix} w_x \\ w_y \\ w_z \end{Bmatrix}$$

Then, the fixed-end force vector in the local frame of reference is:

$$\underline{fef'}_2 = \begin{Bmatrix} -\dfrac{w'_x L}{2} \\ -\dfrac{w'_y L}{2} \\ -\dfrac{w'_z L}{2} \\ 0 \\ \dfrac{w'_z L^2}{12} \\ -\dfrac{w'_y L^2}{12} \\ -\dfrac{w'_x L}{2} \\ -\dfrac{w'_y L}{2} \\ -\dfrac{w'_z L}{2} \\ 0 \\ -\dfrac{w'_z L^2}{12} \\ \dfrac{w'_y L^2}{12} \end{Bmatrix}$$

<u>Wind load</u>

The application of wind load is a bit different in that it is applied to the projected length of the element. Components of wind load on projected length, in the global frame of reference, $(w_{xproj}, w_{yproj}, w_{zproj})$, are provided, with units of force per unit length.

First, compute the projected length of the element on the various planes.

$$x_{proj} = L \cdot \sqrt{m^2 + n^2}$$
$$y_{proj} = L \cdot \sqrt{l^2 + n^2}$$
$$z_{proj} = L \cdot \sqrt{l^2 + m^2}$$

Next, find the total global force components acting on the element.

$$F_x = w_{xproj} \cdot x_{proj}$$
$$F_y = w_{yproj} \cdot y_{proj}$$
$$F_z = w_{zproj} \cdot z_{proj}$$

Transform to local components and divide by element length to get distributed force.

$$w'_x = \frac{\left(l \cdot F_x + m \cdot F_y + n \cdot F_z\right)}{L}$$
$$w'_y = \frac{\left(l_2 \cdot F_x + m_2 \cdot F_y + n_2 \cdot F_z\right)}{L}$$
$$w'_z = \frac{\left(l_3 \cdot F_x + m_3 \cdot F_y + n_3 \cdot F_z\right)}{L}$$

Then, as with the other distributed loads, the fixed-end force vector in the local frame of reference is:

$$\underline{fef'}_3 = \begin{Bmatrix} -\frac{w'_x L}{2} \\ -\frac{w'_y L}{2} \\ -\frac{w'_z L}{2} \\ 0 \\ \frac{w'_z L^2}{12} \\ -\frac{w'_y L^2}{12} \\ -\frac{w'_x L}{2} \\ -\frac{w'_y L}{2} \\ -\frac{w'_z L}{2} \\ 0 \\ -\frac{w'_z L^2}{12} \\ \frac{w'_y L^2}{12} \end{Bmatrix}$$

Effect of temperature change

The coefficient of thermal expansion, $a$, and the change in temperature, *DT*, are provided. The only effect considered is that of constant distribution of temperature through the element cross section. The fixed-end force vector in the local frame of reference is:

$$\underline{fef'}_4 = AEa\Delta T\begin{Bmatrix} 1 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ -1 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{Bmatrix}$$

The total fixed-end force vector for the element in the local frame of reference is:

$$\underline{fef'}_{member} = fef_1' + l\left(\underline{fef'}_2 + \underline{fef'}_3 + \underline{fef'}_4\right)$$

In global components,

$$\underline{fef}_{member} = \underline{\underline{T}}^T\ \underline{fef'}_{member}$$

*Tapered element*

As opposed to prismatic beams, which have well known expressions for fixed-end forces, these values for a tapered element must be obtained through a rather involved process. Basically, a statically determinate, simply-supported beam is considered with the distributed load on it. Using statics, reactions, moments, and shears are determined for both the applied load and unit moments at the ends. From virtual work principles, end rotations can be computed. Then, using the element stiffness matrix already defined, member forces that are required to obtain a state of zero end rotations are computed. Using the resulting end shears to modify the reactions from the simply-supported case, fixed-end forces are obtained.

A number of section types can be specified. Section properties, and the resulting weight, pressure, and ice loads, may be defined at specific points along the element. Values at the ends and center point of each element are used to compute the distributed load value. Note that, because the cross section of the tapered element is given by a radius, wind load is given in units of pressure rather than force per length. Define:

$q_0$ = distributed load at end 1

$q_1$ = distributed load at end 2

$q_{0.5}$ = distributed load at the center.

For the general case, we will assume a lateral force distribution of:

$$q = a_1 + a_2\left(\frac{x}{L}\right) + a_3\left(\frac{x}{L}\right)^2$$

Then,

$$\begin{aligned} a_1 &= q_0 \\ a_2 &= -q_1 + 4q_{0.5} - 3q_0 \\ a_3 &= 2q_1 - 4q_{0.5} + 2q_0 \end{aligned}$$

Obviously, if the distribution is constant, $a_2$ and $a_3$ are zero. If the distribution is linear, $a_3$ is zero. The total force for the element may be computed as:

$$F = \int_0^L \left[ a_1 + a_2\left(\frac{x}{L}\right) + a_3\left(\frac{x}{L}\right)^2 \right] dx = a_1 L + a_2\frac{L}{2} + a_3\frac{L}{3}$$

The location of the centroid of the load must be determined. Compute the integral:

$$I = \int_0^L \left[ a_1 + a_2\left(\frac{x}{L}\right) + a_3\left(\frac{x}{L}\right)^2 \right] x dx = a_1\frac{L^2}{2} + a_2\frac{L^2}{3} + a_3\frac{L^2}{4}$$

Then, the centroid may be computed as:

$$\bar{x} = L\left( \frac{\frac{a_1}{2} + \frac{a_2}{3} + \frac{a_3}{4}}{a_1 + \frac{a_2}{2} + \frac{a_3}{3}} \right)$$

Reactions may now be found:

$$\begin{aligned} R_1 &= F\left(1 - \frac{\bar{x}}{L}\right) \\ R_2 &= F\left(\frac{\bar{x}}{L}\right) \end{aligned}$$

where $R_1$ and $R_2$ are reactions for the simply-supported element at ends *1* and *2*, respectively. Knowing reactions, moment and shear may be determined at any point, *x*, along the length of the element.

The objective is to obtain the element end rotations using the Unit Load Method, which is based on Virtual Work principles. In general,

$$q_1 = \int_0^L \frac{m_1 M}{EI} dx \text{ , } q_2 = \int_0^L \frac{m_2 M}{EI} dx$$

where $m_1$ and $m_2$ are moment functions from unit couples applied at ends *1* and *2*, respectively, and $M$ is the moment function from the applied member load. Moment of inertia, $I$, is also a function of $x$.

$$m_1 = \frac{x}{L} - 1 \text{, } m_2 = \frac{x}{L}$$

The integrations to obtain end rotations are carried out using the trapezoidal rule. In a similar way, axial deflections are computed from applied axial element loads. With the deflections of the simply-supported member known, fixed-end forces are obtained by forcing the deflections back to zero. This is done by changing the sign of the deflections and multiplying them by the known element stiffness matrix.

## Element Forces

Once the displacements for the nodes have been determined, element forces must be computed. The manner in which they are computed varies depending on which type of analysis is being performed.

*First-order analysis*

Because the structural properties are assumed to remain unchanged in first-order analysis, the computation of element force is quite straightforward. First, the node displacements, with components in the global frame of reference, are gathered.

$$\underline{u}_{member} = \begin{Bmatrix} u_A \\ v_A \\ w_A \\ q_{xA} \\ q_{yA} \\ q_{zA} \\ u_B \\ v_B \\ w_B \\ q_{xB} \\ q_{yB} \\ q_{zB} \end{Bmatrix}$$

where *A* and *B* refer to the ends of the element, shown in Figure 5. Then, the global element force components are:

$$\underline{f}_{member} = \underline{\underline{k}}\,\underline{u}_{member} + \underline{fef}_{member}$$

Local axial force components are:

$$\underline{f'}_{member} = \underline{\underline{T}}\,\underline{f}_{member}$$

### *P-D analysis*

The procedure for computing element forces is identical to that for first-order analysis except that $\underline{\underline{k}}_t$ is used instead of $\underline{\underline{k}}$.

### *Second-order analysis*

For second-order analysis, accuracy of the solution is governed primarily by the computation of element forces, used to check equilibrium. Because the updated frame of reference is used to define components of force, moment, displacement, and rotation, its accuracy is essential. While truss elements are assumed to remain straight during deformation and, hence, element orientation depends solely on the location of nodes, frame elements are subject to deformations. That is, although they are assumed to be initially straight, member deformation (i.e., the development of curvature) causes the ends to rotate independently. Thus, each end must have its own frame of reference and, if rotations are assumed to be finite, the orientations must be updated by the theory of finite rotations. The development shown here is taken from Yang and Kuo (1994).

#### <u>Reference axes of element nodes</u>

A set of orthogonal axes is attached to each node of the frame element, originally assumed to be parallel to the global reference axes. At the start of the analysis, denoted as step 1, the axes for node *a* are denoted as ${}^{1}\underline{x}_{a}$, ${}^{1}\underline{h}_{a}$, and ${}^{1}\underline{z}_{a}$.

These axes are updated in accordance with the theory of finite rotations. For any step in the solution process, nodal rotation is represented by a vector:

$$\underline{\Delta q_a} = \Delta q_x \underline{i} + \Delta q_y \underline{j} + \Delta q_z \underline{k}$$

where the components are the rotations for the degrees of freedom, as the structure deforms from step 1 to step 2. Its magnitude and unit direction vector are:

$$f_a = \left|\underline{\Delta q_a}\right| = \sqrt{\Delta q_x^2 + \Delta q_y^2 + \Delta q_z^2}$$

$$\underline{n}_a = \frac{\Delta q_a}{f_a}$$

Then, the updated orientation of the axes at node *a* is:

$$\begin{aligned}
{}^2\underline{x}_a &= \cos f_a \cdot {}^1\underline{x}_a + \sin f_a \cdot \left(\underline{n}_a \times {}^1\underline{x}_a\right) + \left(1 - \cos f_a\right)\left(\underline{n}_a \cdot {}^1\underline{x}_a\right)\underline{n}_a \\
{}^2\underline{h}_a &= \cos f_a \cdot {}^1\underline{h}_a + \sin f_a \cdot \left(\underline{n}_a \times {}^1\underline{h}_a\right) + \left(1 - \cos f_a\right)\left(\underline{n}_a \cdot {}^1\underline{h}_a\right)\underline{n}_a \\
{}^2\underline{z}_a &= \cos f_a \cdot {}^1\underline{z}_a + \sin f_a \cdot \left(\underline{n}_a \times {}^1\underline{z}_a\right) + \left(1 - \cos f_a\right)\left(\underline{n}_a \cdot {}^1\underline{z}_a\right)\underline{n}_a
\end{aligned}$$

### Reference axes for the element

A similar procedure to that for the rotation of nodal axes is needed to update the orientation of the element. Here, the axis of the element is still assumed to run from the start node to the end node (as updated with current displacements), but the orientation of the strong and weak axes of bending also need to be updated. The issue is that the rotation of the nodes at each end is different.

The procedure employed was to base the element orientation on that of the point at midspan. The components of the rotation vectors at the nodes are averaged, and the updated orientation of the element is then obtained using the theory of finite rotations.

### Natural deformation

In order to calculate the change in element forces as it goes from one position to the next, element deformation must be separated from rigid-body motion. In this way, existing element forces are assumed to rotate with the member into the updated configuration. The change in element length and the relative changes in angle at the ends, which is here called *natural deformation*, cause stretch and curvature in the element which, in turn, lead to change in element forces.

The components of rotation, $q_{xa}$, $q_{ya}$, and $q_{za}$, for the node at the *a*-end and $q_{xb}$, $q_{yb}$, and $q_{zb}$ for the node at the *b*-end, must be obtained in the local frame of reference. To get these, a set of vectors in the element local coordinate system that orient the nodes in both the old and the updated configurations must be defined. The notation is:

$${}^{k}_{k-1}\underline{\underline{p}}_a = \left\lfloor {}^{k}_{k-1}\underline{a}_a \quad {}^{k}_{k-1}\underline{b}_a \quad {}^{k}_{k-1}\underline{g}_a \right\rfloor$$

where ${}^{k}_{k-1}\underline{\underline{p}}_a$ is the updated array of local unit vectors for end *a* at step *k* (Note that, while the usual transformation matrix is composed of the components of unit base vectors arranged as rows, the unit base vectors are here arranged as columns.), defined with respect to the element frame of reference at step *k* – 1. These vectors are obtained from the global nodal orientations above via the two step transformation:

$$
\begin{aligned}
{}^{k}_{0}\underline{\underline{p}}_{a} &= {}^{k}\underline{\underline{T}}\left[{}^{k}\underline{x}_{a} \quad {}^{k}\underline{h}_{a} \quad {}^{k}\underline{z}_{a}\right] \\
{}^{k}_{k-1}\underline{\underline{p}}_{a} &= \left[{}^{k-1}_{0}\underline{a}_{a} \quad {}^{k-1}_{0}\underline{b}_{a} \quad {}^{k-1}_{0}\underline{g}_{a}\right]^{T} \cdot {}^{k}_{0}\underline{p}_{a}
\end{aligned}
$$

Note that ${}^{k}\underline{\underline{T}}$ is the element transformation matrix, updated to step *k*. The result is the orientation of the node at end *a* of the element, in the frame of reference of the previous step. A similar process is used for the *b*-end of the element.

Knowing the orientation of the nodes for step *k* in the local reference frame of the previous step, *k* – 1, the direction of the resultant rotation vector, $\underline{n}$, and the angle of rotation, $f$, can be obtained for ends *a* and *b*. Define:

$$
{}^{k}_{k-1}l = \sqrt{\left({}^{k}_{k-1}g_{2} - {}^{k}_{k-1}b_{3}\right)^{2} + \left({}^{k}_{k-1}a_{3} - {}^{k}_{k-1}g_{1}\right)^{2} + \left({}^{k}_{k-1}b_{1} - {}^{k}_{k-1}a_{2}\right)^{2}}
$$

Then,

$$
{}^{k}_{k-1}f = \sin^{-1}\left(\frac{{}^{k}_{k-1}l}{2}\right)
$$

and

$$
\begin{aligned}
{}^{k}_{k-1}n_{1} &= -\frac{{}^{k}_{k-1}g_{2} - {}^{k}_{k-1}b_{3}}{{}^{k}_{k-1}l} \\
{}^{k}_{k-1}n_{2} &= -\frac{{}^{k}_{k-1}a_{3} - {}^{k}_{k-1}g_{1}}{{}^{k}_{k-1}l} \\
{}^{k}_{k-1}n_{3} &= -\frac{{}^{k}_{k-1}b_{1} - {}^{k}_{k-1}a_{2}}{{}^{k}_{k-1}l}
\end{aligned}
$$

The vectors, ${}^{k}_{k-1}\underline{n}_{a}$ and ${}^{k}_{k-1}\underline{n}_{b}$, have components defined with respect to the local nodal axis systems for the previous step, *k* – 1. They must be transformed into the element local frame of reference.

$$
\begin{aligned}
{}^{k}\underline{n}'_{a} &= \left[{}^{k-1}_{0}\underline{a}_{a} \quad {}^{k-1}_{0}\underline{b}_{a} \quad {}^{k-1}_{0}\underline{g}_{a}\right] \cdot {}^{k}_{k-1}\underline{n}_{a} \\
{}^{k}\underline{n}'_{b} &= \left[{}^{k-1}_{0}\underline{a}_{b} \quad {}^{k-1}_{0}\underline{b}_{b} \quad {}^{k-1}_{0}\underline{g}_{b}\right] \cdot {}^{k}_{k-1}\underline{n}_{b}
\end{aligned}
$$

Finally, the natural rotations of the nodes from step *k* – 1 to step *k* are:

$$\begin{Bmatrix} q_{xa} \\ q_{ya} \\ q_{za} \end{Bmatrix} = f_a \cdot {}^{k}\underline{n}'_a$$

$$\begin{Bmatrix} q_{xb} \\ q_{yb} \\ q_{zb} \end{Bmatrix} = f_b \cdot {}^{k}\underline{n}'_b$$

In the updated element frame of reference, the only other natural deformation is element stretch, denoted as $U_b$, with $U_a$ assumed to be zero.

$$U_b = {}^{k}L - {}^{k-1}L$$

The vector of natural deformations is constructed as:

$$\underline{u}_{nat} = \begin{Bmatrix} 0 \\ 0 \\ 0 \\ q_{xa} \\ q_{ya} \\ q_{za} \\ U_b \\ 0 \\ 0 \\ q_{xb} \\ q_{yb} \\ q_{zb} \end{Bmatrix}$$

Incremental local element forces

The local force increments are obtained by multiplying the natural deformations by the local element tangent stiffness matrix and adding the result to the existing local forces. However, the joint moment matrix, $\underline{\underline{k}}_j$, is actually asymmetric. It was made symmetric during the formation of $\underline{\underline{k}}_t$ because the structure of the solver required it. Slight modifications of the stiffness matrix in the predictor phase of the analysis do not harm the solution because the predictor is not fully up to date and, therefore, not expected to be fully accurate. In the equilibrium checking phase, however, maximum accuracy in the most updated configuration is required.

In the solution process, then, the element local tangent stiffness matrix, ${}^{k}\underline{\underline{k'}}_{t}$ is generated in its most updated configuration, and then modified by the addition of the following matrix:

$$
{}^{k}\underline{\underline{k'}}_{j-\mathrm{mod}} = \begin{bmatrix}
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & \frac{{}^{k}f_{6}}{2} & -\frac{{}^{k}f_{5}}{2} & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & \frac{3{}^{k}f_{6}}{2} & 0 & -\frac{{}^{k}f_{4}}{2} & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & -\frac{3{}^{k}f_{5}}{2} & \frac{{}^{k}f_{4}}{2} & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & \frac{{}^{k}f_{12}}{2} & -\frac{{}^{k}f_{11}}{2} \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & \frac{3{}^{k}f_{12}}{2} & 0 & -\frac{{}^{k}f_{10}}{2} \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & -\frac{3{}^{k}f_{11}}{2} & \frac{{}^{k}f_{10}}{2} & 0
\end{bmatrix}
$$

The element forces used in the tangent stiffness matrix are the most current ones available. Then,

$$
{}^{k}\underline{f'}_{elastic} = {}^{k}\underline{\underline{k'}}_{t} \cdot \underline{u}_{nat} + {}^{k-1}\underline{f'}_{elastic}
$$

Finally, updated fixed-end forces are added.

$$
{}^{k}\underline{f'} = {}^{k}\underline{f'}_{elastic} + {}^{k}\underline{fef'}_{member}
$$

# Appendix C
# Matrices for the Spring Element

## Stiffness matrices

The first-order stiffness matrix for the spring element in the local frame of reference is the same as that for the prismatic frame element except that stiffness coefficients for axial behavior and torsion have been replaced with single constants, $k_A$ and $k_T$, respectively and a rotational spring has been inserted at the *b*-end. All other aspects of transformation, the tangent stiffness matrix, finite rotations, and the computation of element forces are unchanged.

$$\underline{\underline{k}}' = \begin{bmatrix}
k_A & 0 & 0 & 0 & 0 & 0 & -k_A & 0 & 0 & 0 & 0 & 0 \\
0 & \frac{12EI_3}{L^3} & 0 & 0 & 0 & \frac{6EI_3}{L^2} & 0 & -\frac{12EI_3}{L^3} & 0 & 0 & 0 & \frac{6EI_3}{L^2} \\
0 & 0 & \frac{12EI_2}{L^3} & 0 & -\frac{6EI_2}{L^2} & 0 & 0 & 0 & -\frac{12EI_2}{L^3} & 0 & -\frac{6EI_2}{L^2} & 0 \\
0 & 0 & 0 & k_T & 0 & 0 & 0 & 0 & 0 & -k_T & 0 & 0 \\
0 & 0 & -\frac{6EI_2}{L^2} & 0 & \frac{4EI_2}{L} & 0 & 0 & 0 & \frac{6EI_2}{L^2} & 0 & \frac{2EI_2}{L} & 0 \\
0 & \frac{6EI_3}{L^2} & 0 & 0 & 0 & \frac{4EI_3}{L} & 0 & -\frac{6EI_3}{L^2} & 0 & 0 & 0 & \frac{2EI_3}{L} \\
-k_A & 0 & 0 & 0 & 0 & 0 & k_A & 0 & 0 & 0 & 0 & 0 \\
0 & -\frac{12EI_3}{L^3} & 0 & 0 & 0 & -\frac{6EI_3}{L^2} & 0 & \frac{12EI_3}{L^3} & 0 & 0 & 0 & -\frac{6EI_3}{L^2} \\
0 & 0 & -\frac{12EI_2}{L^3} & 0 & \frac{6EI_2}{L^2} & 0 & 0 & 0 & \frac{12EI_2}{L^3} & 0 & \frac{6EI_2}{L^2} & 0 \\
0 & 0 & 0 & -k_T & 0 & 0 & 0 & 0 & 0 & k_T & 0 & 0 \\
0 & 0 & -\frac{6EI_2}{L^2} & 0 & \frac{2EI_2}{L} & 0 & 0 & 0 & \frac{6EI_2}{L^2} & 0 & \frac{4EI_2}{L} & 0 \\
0 & \frac{6EI_3}{L^2} & 0 & 0 & 0 & \frac{2EI_3}{L} & 0 & -\frac{6EI_3}{L^2} & 0 & 0 & 0 & \frac{4EI_3}{L}
\end{bmatrix}$$

Internal spring

The element is assumed to have a rotational spring at the *b*-end, as shown. To properly include its stiffness, an internal node is assumed, infinitesimally close to the *b*-end node. The beam element and the spring element are assembled in the usual way to form a 3-node element. Then, the internal node is removed using static condensation. In the following derivations, only bending stiffness is considered.

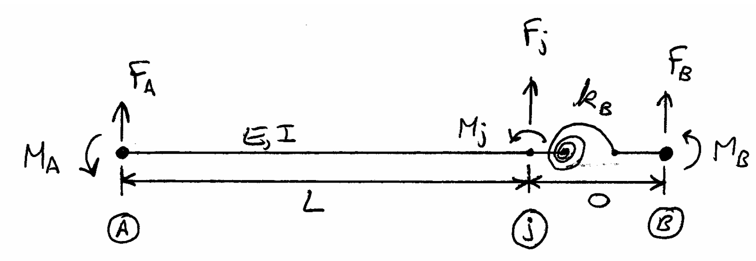


Figure C-1. Schematic for bending behavior of the spring element.

The stiffness matrix for the rotational spring is:

$$\begin{Bmatrix} M_j \\ M_B \end{Bmatrix} = k_b \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} q_j \\ q_B \end{Bmatrix}$$

The stiffness matrix for the beam element is:

$$\begin{Bmatrix} F_A \\ M_A \\ F_j \\ M_j \end{Bmatrix} = \begin{bmatrix} \frac{12EI}{L^3} & \frac{6EI}{L^2} & -\frac{12EI}{L^3} & \frac{6EI}{L^2} \\ \frac{6EI}{L^2} & \frac{4EI}{L} & -\frac{6EI}{L^2} & \frac{2EI}{L} \\ -\frac{12EI}{L^3} & -\frac{6EI}{L^2} & \frac{12EI}{L^3} & -\frac{6EI}{L^2} \\ \frac{6EI}{L^2} & \frac{2EI}{L} & -\frac{6EI}{L^2} & \frac{4EI}{L} \end{bmatrix} \begin{Bmatrix} v_A \\ q_A \\ v_j \\ q_j \end{Bmatrix}$$

For the assumption that $F_j = F_B$ and $v_j = v_B$, we can assemble the two elements as follows:

$$\begin{Bmatrix} F_A \\ M_A \\ F_B \\ M_B \\ M_j \end{Bmatrix} = \begin{bmatrix} \frac{12EI}{L^3} & \frac{6EI}{L^2} & -\frac{12EI}{L^3} & 0 & \frac{6EI}{L^2} \\ \frac{6EI}{L^2} & \frac{4EI}{L} & -\frac{6EI}{L^2} & 0 & \frac{2EI}{L} \\ -\frac{12EI}{L^3} & -\frac{6EI}{L^2} & \frac{12EI}{L^3} & 0 & -\frac{6EI}{L^2} \\ 0 & 0 & 0 & k_b & -k_b \\ \frac{6EI}{L^2} & \frac{2EI}{L} & -\frac{6EI}{L^2} & -k_b & \frac{4EI}{L}+k_b \end{bmatrix} \begin{Bmatrix} v_A \\ q_A \\ v_B \\ q_B \\ q_j \end{Bmatrix}$$

Static condensation procedures are employed to remove the last row and column, which pertain to the degrees of freedom of the internal node, *j*. The remaining stiffness coefficients are those that contribute to degrees of freedom 2, 6, 8, and 12 for bending in the local *1 – 2* plane, and to degrees of freedom 3, 5, 9, and 11 for bending in the local *1 – 3* plane.