FINITE ELEMENT MODELING

IN ENGINEERING PRACTICE

Includes Examples with ALGOR®

Constantine C. Spyrakos

Table of Contents

	PREFACE	*i*
Chapter 1	**BASICS OF FINITE ELEMENT ANALYSIS**	
1.1	Introduction	*1*
1.2	General Description of the Finite Element Method and Steps of Finite Element Analysis	*2*
1.3	Introductory Example	*4*
	Static Analysis	*5*
	Dynamic Analysis	*11*
Chapter 2	**STRESSES AND FAILURE CRITERIA**	
2.1	Concept of Stress	*17*
2.2	Principal Stresses	*18*
2.3	Stresses in Trusses and Beams	*19*
2.4	Plane Stress	*21*
2.5	Plane Strain	*23*
2.6	Axisymmetric Solids	*24*
2.7	Plates and Shells	*26*
2.8	Isotropy - Orthotropy - Anisotropy	*27*
2.9	Failure Criteria	*29*
	Tresca Yield Criterion	*31*
	Von Mises Yield Criterion	*32*
	Maximum Normal Stress Criterion	*32*
Chapter 3	**FINITE ELEMENT MODELING FUNDAMENTALS**	
3.1	Modeling Considerations	*33*
3.2	Types of Finite Elements	*34*
	Truss Elements	*34*
	Beam Elements	*35*

Two-Dimensional Solid Elasticity Elements 38
Plane Stress Elements 38
Plane Strain Elements 40
Axisymmetric Elements 41
Three-Dimensional Plane Stress or Membrane Elements 42
Plate and Shell Elements 43
Three-Dimensional Solid Elasticity or Brick Elements 45
Tetrahedral and Hexahedral Elements 46
Boundary Elements 47
Gap Elements 48
3.3 Selection of Element Types 49
Combination of Different Types of Elements 51
3.4 Modeling Guidelines 53
General Recommendations 53
Aspect Ratio 54
Skewed Elements 55
Geometric Discontinuities 55
Material Discontinuities 58
Abrupt Changes in Stiffness 58
Abrupt Changes in Loads 58
3.5 Application of Boundary Conditions 59
3.6 Application of Loads 60
3.7 Stiff and Rigid Structural Parts 63
3.8 Skewed Boundaries 64
3.9 Symmetry in Static and Dynamic Analysis 65
Static Analysis 65
Dynamic Analysis 70
3.10 Recommendations for Evaluating Stresses 73

Chapter 4 CONCEPTS OF DYNAMIC ANALYSIS

4.1 Introduction 75
4.2 Free Vibration of Single-Degree-of-Freedom System (SDOF) 76
4.3 Natural Frequency and Period 78
4.4 Free Vibration of Damped SDOF System 80
4.5 Undamped Excitation to Harmonic External Forces 84
4.6 Damped Excitation to Harmonic Forces 86
4.6.1 Transient and Steady-State Response 86
4.6.2 Dynamic Magnification Factor 87
4.6.3 Effect of Loading Duration on Response Amplitude 89
4.7 Excitation to Harmonic Ground Motion 91

4.8 Response to General Excitation 92
4.9 The Concept of Response Spectrum 94
4.10 Random Vibration Analysis 98
4.10.1 Introduction 98
4.10.2 Probability Distribution 100
4.10.3 Power Spectral Density and Root Mean Square Response 103
4.11 Fourier Series and Fourier Transform 107
4.11.1 Introduction and Definitions 107
4.11.2 Transfer Function 110
4.11.3 Root Mean Square Response of SDOF System 112
4.12 Damping in Structural Systems 113
4.12.1 Introduction 113
4.12.2 Classification of Damping 114
4.12.3 Equivalent Viscous Damping 119

Chapter 5 DYNAMIC ANALYSIS WITH FINITE ELEMENTS

5.1 Introduction: Recommended Approach 121
5.2 Equations of Motion 122
5.3 Mass: Lumped and Distributed 124
5.4 Modal Analysis or Mode Shape Analysis: Definitions and Physical Interpretation 125
Practical Considerations for Modal Analysis 136
5.5 Modal Analysis with Load Stiffening 137
5.6 Sturm Frequency Check 138
5.7 Cut-off Frequency 138
5.8 Rigid Body Modes 139
5.9 How to Incorporate Damping in FEM 141
5.10 Time History Modal Superposition 142
5.11 Practical Considerations for Time History Modal Superposition 148
5.12 Time History Direct Integration 150
5.13 Time History: Modal Superposition versus Direct Integration 151
5.14 Frequency Response Analysis 152
5.15 Response Spectrum Analysis 158
5.16 Modal Combination Methods 163
5.17 Random Vibration Analysis 166

Chapter 6 STATIC ANALYSIS: NUMERICAL EXAMPLES

6.1 Introduction 171
6.2 Analysis of Short Beam 172

6.3 Analysis with Algor 174
6.4 Interpretation of Graphical Results 181
6.4.1 Displacement Results 181
6.4.2 Stress Results 183
6.4.3 Von Mises, Tresca, and Principal Stresses 184
6.4.4 Reaction Forces 185
6.4.5 Components of the S-tensor 186
6.4.6 Precision 188
6.5 Interpretation of Numerical Results 188
6.5.1 Displacement Output 189
6.5.2 Stress Output 193
6.5.3 Reaction Force Output 195
6.6 Accuracy and Convergence Studies 198
6.7 Analysis of Plate Structure 201
6.7.1 Analysis with Algor 202
6.7.2 Interpretation of Graphical Results 206
6.7.3 Interpretation of Numerical Results 210

Chapter 7 MODAL ANALYSIS: NUMERICAL EXAMPLES

7.1 Introduction 215
7.2 Introductory Example 218
7.2.1 Analysis with Algor 219
7.2.2 Interpretation of Results 222
7.2.3 Numerical Accuracy and Comparisons 226
7.3 Symmetric Structures 227
7.3.1 Beam Structure 227
7.3.2 Analysis with Algor 230
7.3.3 Plate Structure 230
7.3.4 Analysis with Algor 233
7.4 Concentrated Mass and Stiffness 235
7.4.1 Analysis with Algor 236
7.5 Frame with Concentrated Masses 238
7.5.1 Analysis with Algor 239
7.6 System with Rigid Body Modes 243
7.6.1 Analysis with Algor 244
7.7 Load Stiffened Structure 246
7.7.1 Analysis with Algor 249

Chapter 8 TIME HISTORY ANALYSIS: NUMERICAL EXAMPLES

8.1 Introduction 251

8.2 Time History Modal Superposition: Introductory Example *251*
8.2.1 Analysis with Algor *254*
8.2.2 Interpretation of Graphical Results *257*
8.2.3 Numerical Results *258*
8.3 Use of Symmetry in Time History Modal Superposition *261*
8.3.1 Loads on Axis of Symmetry *261*
8.3.2 Use of Symmetry for Non-symmetrically Loaded Structures *262*
8.4 Time History Direct Integration *264*
8.4.1 Numerical Example *265*
8.4.2 Analysis with Algor *266*
8.4.3 Numerical Results *271*

Chapter 9 FREQUENCY RESPONSE, RESPONSE SPECTRUM, AND RANDOM VIBRATION ANALYSIS: NUMERICAL EXAMPLES

9.1 Frequency Response Analysis: Numerical Examples *275*
9.1.1 System Subjected to Harmonic Loads *276*
9.1.2 Analysis with Algor *277*
9.1.3 Graphical Results *279*
9.1.4 Numerical Results *281*
9.2 Response Spectrum Analysis: Numerical Examples *287*
9.2.1 System Subjected to Response Spectra of a Harmonic Ground Excitation *288*
9.2.2 Analysis with Algor *290*
9.2.3 Graphical Results *292*
9.2.4 Numerical Results *294*
9.2.5 System Subjected to Design Response Spectra *298*
9.2.6 Analysis with Algor *299*
9.2.7 Interpretation of Results *301*
9.3 Random Vibration Analysis: Numerical Examples *304*
9.3.1 Analysis with Algor *304*
9.3.2 Interpretation of Results *306*

BIBLIOGRAPHY *313*

INDEX *317*

3

Finite Element Modeling Fundamentals

3.1 MODELING CONSIDERATIONS

The objective of finite element analysis is to accurately determine the response of a system modeled with finite elements and subjected to given loads. In creating a finite element model, always keep in mind that we are developing a model which is an idealization of the real physical system. With very few exceptions, such as the static analysis of simple truss, frame, and membrane systems, finite element analysis does not provide "exact" answers. However, with proper modeling, it yields accurate solutions. When an analytical formulation of a problem is difficult to develop, FEM has proven to be one of the most reliable methods to tackle the problem.

In creating a FEM model one should strive for accuracy and computational efficiency. In most cases, use of a complex and very refined model is not justifiable since it most likely provides computational accuracy at the expense of unnecessarily increased processing time. The type and complexity of the model is dependent upon the type of results required. As a general rule, finite element modeling should start with a simple model. The results from the simple model combined with an understanding of the behavior of the system will

help us decide whether and at which part of the model further refinement is needed.

3.2 TYPES OF FINITE ELEMENTS

This section describes several salient features of the most commonly used types of finite elements; namely, *truss, beam, plane stress, plane strain, axisymmetric, membrane, plate, shell, solid or brick, tetrahedral, hexahedral, boundary, and gap* elements. The reader may be aware that several commercial finite element programs have numerous types of finite elements in their library. Nevertheless, the majority of structural and mechanical engineering applications can be solved with the aforementioned *basic element* types. Besides the basic elements, other special types, such as *pipe* and *rigid* elements are available in finite element program libraries and can be used for specialized applications.

Depending on their dimensions, basic elements can be divided into three categories: *line, area, and volume elements*. Truss, beam, and boundary elements are line elements. Plane stress, plane strain, axisymmetric, membrane, plate and shell are area elements. Solid or brick, tetrahedral, and hexahedral are volume elements. Criteria for selecting the appropriate elements for the application are provided in Section 3.3. At this point it should be mentioned that there are considerable differences between proper modeling for static and dynamic analysis. These differences as well as their similarities are discussed in Sections 3.4 through 3.10 and in Section 7.1.

Truss Elements

Truss structures such as mechanical linkages, roof frames, and bridges can be modeled with truss elements. In general, a structural member can be modeled with truss elements if it fulfills the following three requirements:

a. its length is much greater than its width and depth (the loosely defined term *much greater* can be quantified from 8 to 10 times greater for most applications);

b. it is connected with the rest of the structure with hinges that do not allow transfer of moments; and

c. it is loaded with external loads applied only at the joints.

Truss elements can only undertake tension or compression. Thus, the only cross-sectional property that needs to be specified is their axial area. The geometrical profiles of truss elements are identical to beam elements. Figure 3.2.1 shows the geometry and nodal forces of a three-dimensional truss element. As depicted in this figure, a three-dimensional truss element has three degrees-of-freedom per node, that is, three displacements along the global X, Y and Z-axes.

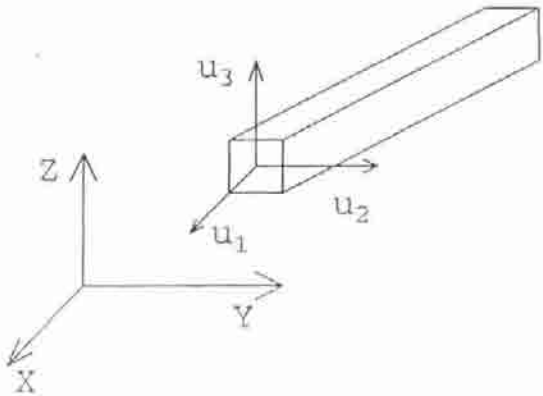

Figure 3.2.1 Three-dimensional truss element.

Beam Elements

Beam elements are probably the most commonly used elements. Besides their obvious application in frame structures, many other systems such as mechanical linkages, piping systems, conduits, and bridge girders can be modeled with beam elements.

For a structural member to be modeled with beam elements, one of its dimensions must be much greater, as a general rule at least 10 times greater, than the other two. Contrary to truss elements, beam elements can undertake shear and moment in addition to tension and compression. The geometry and the displacements/rotations at a node are shown in Fig. 3.2.2. Notice that a three-dimensional beam element has six degrees-of-freedom per node, that is, three displacements along and three rotations about the global X, Y and Z-axes.

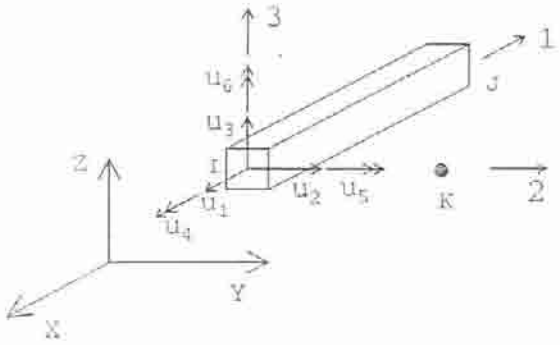

Figure 3.2.2 Three-dimensional beam element.

Common beam profiles include I-sections, T-sections, box, circular, and channel sections. The cross-sectional properties that must be specified for a beam element include the axial area, the shear areas, the torsional resistance, the flexural moments of inertia, and the section moduli. Table 3.2.1 presents the shear areas of several common beam profiles.

Unless the beam cross-sectional area is axisymmetric, e.g., a circular or a square cross section, the orientation of the beam element with respect to the global system XYZ is critical. Most finite element programs orient a beam element with the aid of an additional node "K," which together with the two end-nodes "I" and "J," specify the local coordinate system 123, see Fig. 3.2.2. Note that the 123 and the XYZ systems in Fig. 3.2.2 coincide. In the general case, however, the two systems are not identical.

It is recommended that the appropriate non-zero value for the shear areas is used in the analysis, since accounting for shear deformations improves the solution accuracy. Usually, non-zero values for shear areas must be assigned if $L < 10\ D$, where L and D are the longest and the shortest dimensions of the beam, respectively. A more detailed discussion on the effects of shear on the system response and numerical examples are given in Section 7.2.

The torsional constant is equal to the polar moment of inertia only for circular sections. Use of the wrong torsional constant is a common source of error in frame analysis. The torsional constant for non-circular uniform sections is available in various sources in the literature.

Table 3.2.1

Shear Area of Common Cross Sections

Cross Section	Type	Shear Area
	Rectangular Section b : Width d : Thickness	$\frac{5}{6}$ bd
	Wide Flange Section b : Width t : Thickness of flange	$\frac{5}{3}$ bt
	Wide Flange Section h : Height t : Thickness of web	th
	Thin-Wall Circular Tube Section R : Mean radius t : Thickness	$\frac{\pi Rt}{2}$
	Solid Circular Section R : Radius	$0.9\pi R^2$
	Thin-Wall Rectangular Tube Section d : Width t : Thickness	2td

* Arrows indicate direction of shear forces

Two-Dimensional Solid Elasticity Elements

There are three types of two-dimensional elements:

1. Plane Stress Elements
2. Plane Strain Elements
3. Axisymmetric Elements.

For consistency with the notation in Chapter 2, all two-dimensional elements are defined in the global YZ plane. Their use and a description of their configuration and properties is provided in the following.

Plane Stress Elements

A brief presentation of plane stress as treated by the theory of elasticity is given in Section 2.4. When a structural system is under conditions of plane stress all stress components normal to the YZ plane vanish:

$$\sigma_{xx} = \tau_{yx} = \tau_{zx} = 0 \qquad (2.4.1)$$

A representative example of a plane stress problem is a thin plate loaded along its edges with a uniform load in the YZ plane. Due to the small thickness of the plate, the stresses that develop along its thickness are very small

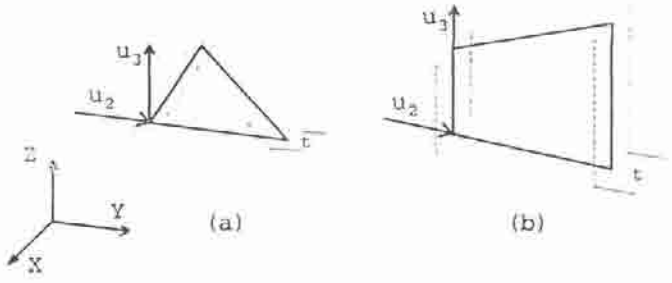

Figure 3.2.3 Two-dimensional plane stress elements: (a) Triangular; (b) Quadrilateral.

compared with the stresses in the YZ plane and they can be ignored in the analysis, see Fig. 2.4.1. Most commonly used plane stress elements are either quadrilateral or triangular. In either case, every node has two translational degrees-of-freedom, as shown in Fig. 3.2.3.

As a rule, prefer quadrilateral over triangular elements for reasons of geometric isotropy. However, it is suggested that triangular elements be used to better model irregular boundaries as well as areas around crack tips, see Fig. 3.2.4.

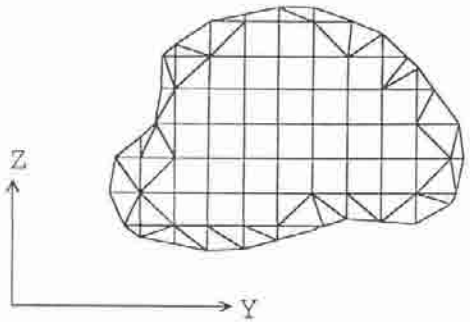

Figure 3.2.4 Quadrilateral and triangular plane stress elements.

Depending on the approximation of stresses within the element, two different plane stress elements are commonly used: elements with a constant stress variation within the element (CST) also known as *compatible*, and elements with a linear stress variation (LST) known as *incompatible*. Incompatible elements are usually effective in modeling regions with significant changes in stress gradient. A more refined mesh is required if compatible elements are used in the analysis. As the best modeling approach that combines the computational efficiency of compatible elements with the computational accuracy of incompatible elements, it has been suggested to use incompatible elements only in those regions of the structure where good solution accuracy is desired, for example, areas of high variation in stress gradient, e.g., around holes and cracks. In practice, however, incompatible elements are used for the whole model. An effort should also be made to keep the *aspect ratio*, i.e., the ratio between the element's longest to shortest dimensions, close to one. For a discussion on the aspect ratio refer to Section 3.4.

In plane stress analysis one should:

a. keep the aspect ratio as close to unity as possible, i.e., strive for either equilateral triangular elements or square elements.

b. keep internal angles close to 90° in quadrilateral elements.

Additional recommendations on proper modeling with plane stress elements are given in Section 3.4.

Plane Strain Elements

As discussed in Section 2.5, a system is considered to be under conditions of plane strain if one of its dimensions, say along the X-axis, is much greater than the other two and is subjected to a load that does not vary along the X-axis, see Fig. 2.5.2. Typical examples are dams, tunnels, retaining walls, but also small scale systems such as rollers and bars compressed by forces normal to their cross section. If the structure fulfills conditions of plane strain, and in addition it is restrained to deform along its longest dimension, say along the X-axis, then the strain ϵ_{xx} is zero, see Fig. 2.5.1. Note, however, that σ_{xx} is not zero. If the structure is not restrained along the X-axis, then the value of ϵ_{xx} is a non-zero constant. The latter case is known as *generalized plane strain*. In either case, however, we can determine the stresses by simply modeling and analyzing a *typical cross section* in the YZ plane that carries the load applied on a unit length along the X-axis.

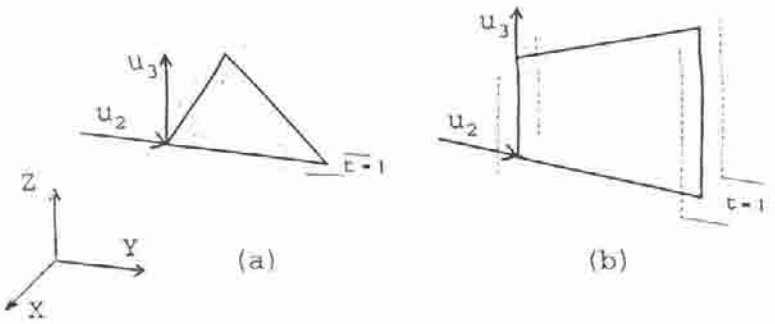

Figure 3.2.5 Plane strain elements: (a) Triangular; (b) Quadrilateral.

The most commonly used plane strain elements are either triangular or quadrilateral with two translational degrees-of-freedom per node, as shown in

Fig. 3.2.5. They have the same geometry and degrees-of-freedom with plane stress elements, their difference being that plane strain elements have unit thickness, while the thickness of plane stress elements is t. Finally, the remarks made on compatible and incompatible as well as the use of a proper aspect ratio for plane stress elements are also valid for plane strain elements.

Axisymmetric Elements

Steel and concrete storage tanks, rotors, shells, nozzles, and nuclear containment vessels are some representative axisymmetric structures. Similar to three-dimensional structures that are under conditions of plane stress or plane strain, axisymmetric structures loaded with axisymmetric loads can be analyzed as two-dimensional systems. In order to analyze an axisymmetric structure, such as a cylinder with thickness t subjected to a uniform internal pressure p, what we model is the intersection of the cylinder with the plane YZ in the positive quadrant of the XYZ Cartesian system drawn with continuous lines in Fig. 3.2.6(b). Note that Z is the axis of load and geometric symmetry and the radial coordinates are specified along the positive Y-axis. Several software do not allow negative Y coordinates, see the Algor Processor Reference Manual. The loading p that is applied to the finite element model shown in Fig. 3.2.6(b) corresponds to a slice of the structure defined by rotating a cross-section of the structure about the Z-axis by a unit angle $\theta = 1$ rad. Axisymmetric quadrilateral and triangular elements have two translational degrees-of-freedom per node, see Figs. 3.2.6(c) and (d).

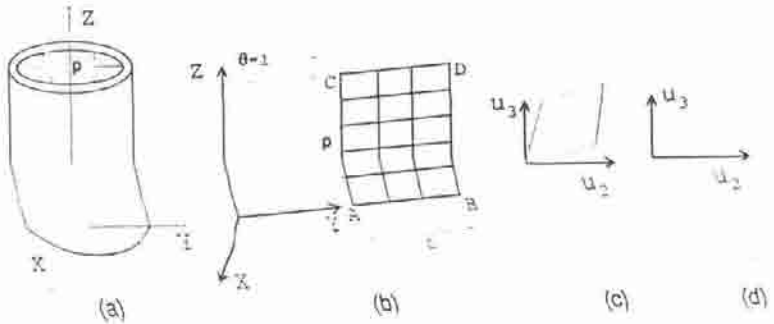

Figure 3.2.6 (a) Cylinder under internal pressure; (b) Axisymmetric model on YZ plane; (c) Quadrilateral element; (d) Triangular element.

Both triangular and quadrilateral elements have proved to be very highly accurate in comparison with classical analytical solutions. The comments made in the discussion on plane stress compatible and incompatible elements are also valid for axisymmetric elements.

Three-Dimensional Plane Stress or Membrane Elements

Examples of structures that can be analyzed with membrane elements include roofs made of fabric-like materials used to cover sports stadiums. Because of their small thickness such structures can develop only in-plane stresses.

Membrane elements can be "viewed" as three-dimensional plane stress elements. The difference is only that membrane elements have three instead of two translational degrees-of-freedom per node, as shown in Fig. 3.2.7. It should be noted that a membrane element does not necessarily lie in the YZ plane which is the case for the two-dimensional plane stress elements shown in Fig. 3.2.3.

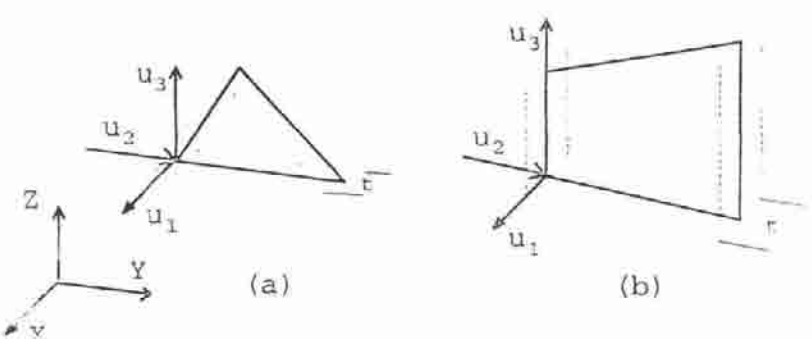

Figure 3.2.7 Membrane elements: (a) Triangular; (b) Quadrilateral.

All comments made regarding compatible and incompatible triangular and quadrilateral plane stress elements are also applicable to membrane elements.

Plate and Shell Elements

Use of plate and shell structures is extensive. It includes architectural structures, containers, airplanes, ships, missiles, and machine parts. Plates are plane, flat surface structures with thickness that is very small compared to their other dimensions, see Section 2.7.

The load-carrying behavior of plates resembles that of beams. In fact, plate behavior can be approximated by a gridwork of beams running in two perpendicular directions on the same plane. The two-dimensional load-carrying action of plates results in light and economical structures. In finite element modeling of a plate, what we model is the *mid-surface* which is a plane parallel to the two plate surfaces that divides the thickness into equal halves.

Modeling of plate structures is usually done with either triangular or quadrilateral plate elements. Each nodal point of either element type possesses five degrees-of-freedom in the element or local system xyz: three displacements along the x, y, and z-axes and two rotations about the x and y-axes. No rotational degree-of-freedom is specified about the direction z normal to the plate element, see Fig. 3.2.8. Nevertheless, a very small value for the rotational stiffness is assigned about the local z-axis to eliminate numerical instability. It

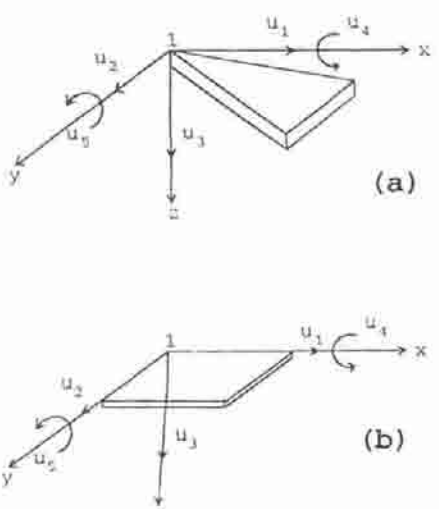

Figure 3.2.8 Plate elements and degrees-of-freedom at node 1; (a) Triangular; (b) Quadrilateral.

should be noted that plate and shell elements used in finite element programs combine bending and membrane actions.

Quadrilateral elements exhibit higher accuracy than triangular elements, and thus they should be preferred. Use of triangular elements should be limited to the part of the structure that is close to irregular boundaries.

Thin shells can be "viewed" as plates with a curved surface. For the definition of thin shell elements refer to Chapter 2. In this text shell elements are understood as flat plate elements used to model curved surfaces. In fact, most commercial finite element programs rely on plate elements to model shells and curved surface structures. As for thin plates, the mid-surfaces of thin shell structures are modeled with plate elements.

Two aspects should be kept in mind when modeling thin shells with plate elements:

a. for good solution accuracy, it is recommended that the plate elements should be approximating the shell surface at angles between 4^0 and 10^0 degrees, see Fig. 3.2.9.

b. lack of a rotational degree-of-freedom about the local z-axis may cause numerical instability. Such case could appear when plate elements intersect at an angle less than 4^0, see Fig. 3.2.10. The numerical instability might not allow processing of the model and could be reported by the processor as an "error message." A remedy to this problem is to use rotational elastic elements with very small stiffness about the axes parallel to the common sides of the elements.

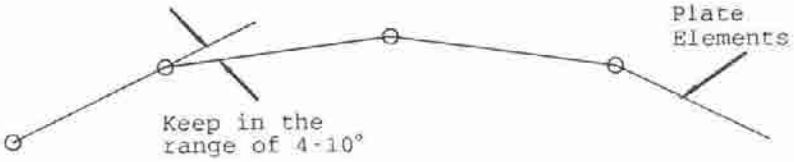

Figure 3.2.9 Modeling of thin shell with plate elements.

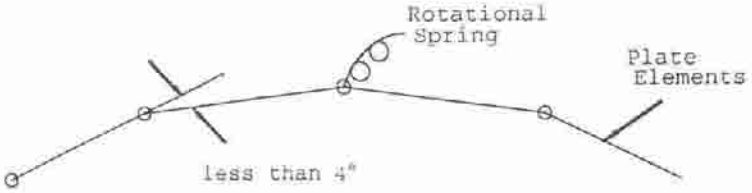

Figure 3.2.10 Small stiffness rotational spring to avoid numerical instability.

Three-Dimensional Solid Elasticity or Brick Elements

Thick plates, thick cylindrical or spherical components, thick joints, and gear housings are a few examples where solid elements can be used to perform a finite element analysis. In general, structures or structural components with a thickness comparable to the other two dimensions can be modeled with brick elements.

Solid elements are three-dimensional elements with three translational degrees-of-freedom per node. They can be viewed as membrane elements that can also account for variation of stresses along the thickness, see Fig. 3.2.11. Nodes are usually introduced at either the intersections of three planes or the mid-sides of the intersection of two planes. An eight-node brick element with the associated degrees-of-freedom at one node is shown in Fig. 3.2.11.

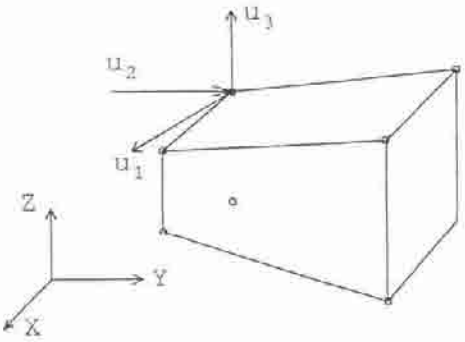

Figure 3.2.11 Brick element and degrees-of-freedom at a node.

The advantage of using solid or brick elements instead of plate or beam elements to model components with large thickness is that solid elements can provide information about the three-dimensional variation of stresses and deformations within the component. Plate and beam elements cannot provide such information and should be used with caution only for a preliminary analysis of systems having comparable sizes in all three dimensions.

When solid elements are interconnected with beam or plate and shell elements, attention should be paid to preserve the rotational degrees-of-freedom at the common nodes. This can be easily achieved by artificially extending the beam and plate elements into the solid elements, see Fig. 3.3.4. The issue of combining different element types is further discussed in Section 3.3.

Tetrahedral and Hexahedral Elements

Instead of brick elements, tetrahedral and hexahedral elements can be used to model three-dimensional structures. The tetrahedron can be viewed as an extension of the triangle in the third dimension, see Fig. 3.2.12, while the hexahedron can be considered as the counterpart of the planar quadrilateral extended in the third dimension. It should be noted that the hexahedral element has the same geometric shape as the eight-node brick element of Fig. 3.2.11. They differ, however, in their theoretical formulation and computational accuracy. Commonly used tetrahedral and hexahedral elements have only three translational degrees-of-freedom per node, see Fig. 3.2.12(a). The accuracy of tetrahedral and hexahedral elements is increased by introducing additional nodes at mid-sides, see Fig. 3.2.12(b).

Automatic mesh generation techniques favor tetrahedral and hexahedral elements over brick elements to create detailed models of three-dimensional systems with complex geometries, e.g., Supergen, Supersurf, Hypergen, and Hexagen of Algor. In practical applications, use of tetrahedral elements requires some caution to develop models without leaving any "holes" in them. In general, hexahedral elements are more accurate than tetrahedral elements and should be preferred. Nevertheless, tetrahedral elements are more versatile since they allow modeling of intricate geometries and facilitate transition from coarsely meshed regions to finely meshed regions in a model. In most cases, combination of tetrahedral and hexahedral provides the optimum mesh. Further elaborations on these elements, including the methods of formulation and comparative studies on their efficiency are given in several references, e.g., Gallagher, Yang, Zienkiewicz and Taylor.

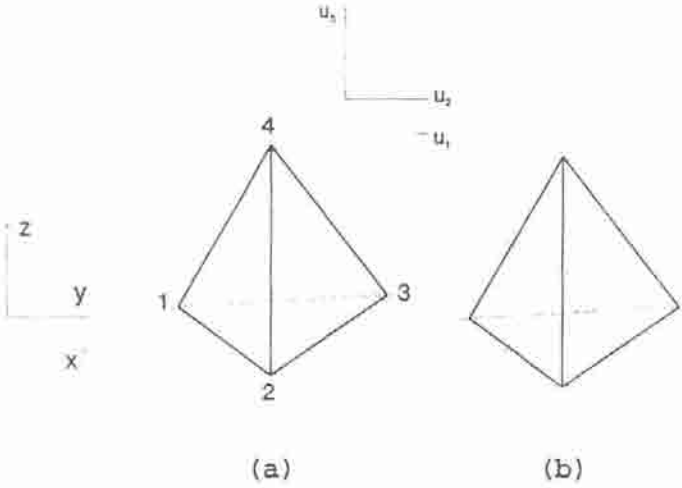

Figure 3.2.12 (a) Four-node tetrahedral; (b) Ten-node tetrahedral.

Boundary Elements

Boundary elements are used to model boundary conditions and links between structural components. As such, they can be useful in either evaluating reactions at rigid, flexible supports, and elastic links or specifying non-zero displacements and rotations to nodes. Boundary elements are two-node elements. The line defined by the two nodes indicates the direction along or about which the reaction (force or moment) is evaluated or the displacement (translation or rotation) is specified, see Fig. 3.2.13.

Boundary elements that are used to obtain reaction forces *(rigid boundary elements)* or specify translational displacements *(displacement boundary elements)* can be considered as truss elements with only one non-zero translational stiffness. When they are employed to either evaluate reaction moments or specify rotations, they resemble beam elements with only one non-zero stiffness, that is, the rotational stiffness about the user specified axis.

Elastic boundary elements are used to model flexible supports and to calculate reactions at skewed boundaries, see Fig. 3.8.1. The stiffness of elastic boundary elements is defined by the user. Caution is needed when one wants to model skewed supports. In this case, the user should employ either translational

or rotational elastic boundary elements and assign moderately high stiffness values to the elastic boundary elements. The process of specifying appropriate stiffness to boundary elements is presented in Section 3.8.

As mentioned in the Section on "Plate and Shell Elements," elastic boundary elements with a small stiffness may also be used in models of thin shell structures, see Fig. 3.2.10.

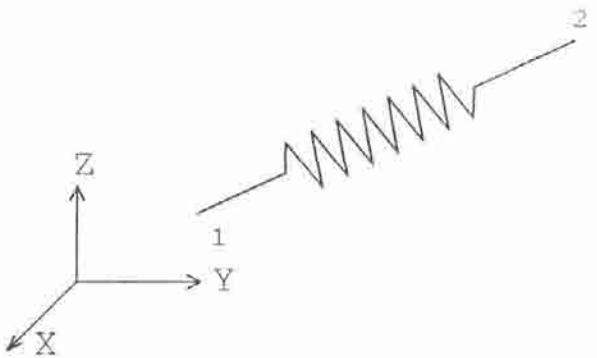

Figure 3.2.13 Boundary element.

Gap Elements

These elements are used to model cables, gaps between structural components, and gaps between a structure and its supports. Gap elements can be either compressive or tensile. Compressive gap elements become active when compressive forces develop on the boundary element after the gap is closed. Similarly, tensile gap elements resist tensile forces after the gap is closed. Figure 3.2.14 shows a frame structure with diagonal cable bracings that can be modeled with gap elements. Since in the braced frame there is no gap between the cables and the other members, the gap elements become active immediately after the load is applied.

It should be mentioned that most structural problems involving the use of gap elements are *nonlinear*. The nonlinearity is attributed to the change of the system's stiffness which occurs when the gap elements are either activated or de-activated. Thus, all the dynamic analysis methods presented for linear problems in Chapters 5, 7, 8, and 9 cannot be applied when modeling requires use of gap elements.

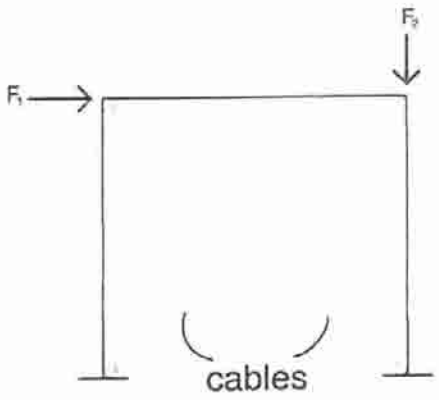

Figure 3.2.14 Braced frame with diagonal cables.

3.3 SELECTION OF ELEMENT TYPES

Before selecting the types of elements to model a structure, we must first draw an outline of the physical system indicating the overall geometry, boundary conditions, loads, and regions of material and geometric discontinuities. The sketch should also include the global coordinate system and the dimensions of the structure.

Next, we should examine whether the size of the model can be reduced. Plane strain, plane stress, and axisymmetry allow reduction of three-dimensional problems to two-dimensional. In addition, the presence of planes of symmetry allows modeling of only a part of the structure. Appropriate use of symmetry in static and dynamic analysis is discussed in Section 3.9.

The outline of the physical system will guide us in selecting the appropriate types of elements. For example, to model transversely or axially loaded components in mechanical, electrical, and civil structures, we could use beam and truss elements. Plane stress elements are appropriate to model in-plane action of plates and short beams. Plane strain elements are usually employed to model retaining walls and long dams. Axisymmetric elements are used to model structures that are rotationally symmetric about an axis and symmetrically or antisymmetrically loaded about the same axis, such as cylinders subjected to internal pressure.

The choice of the element type(s) also depends on the kind of results

requested. For example, the cylinder shown in Fig. 3.3.1 with both ends clamped and subjected to a point load can be modeled in several different ways:

a. A beam model can be used if we are interested in an approximate calculation of the deflections.

b. A model using shell/plate elements can be used if an accurate calculation of the stresses is our objective.

Choice between plate or solid elements is not always straightforward. As a general rule, use plate elements to model non-axisymmetric thin shell and plate structures, and brick, tetrahedral, and hexahedral elements to model thick plate and shell structures. Support movements and elastic supports can be modeled with displacement and elastic boundary elements, respectively. Gap elements are appropriate to model cables and gaps between structures and structural components.

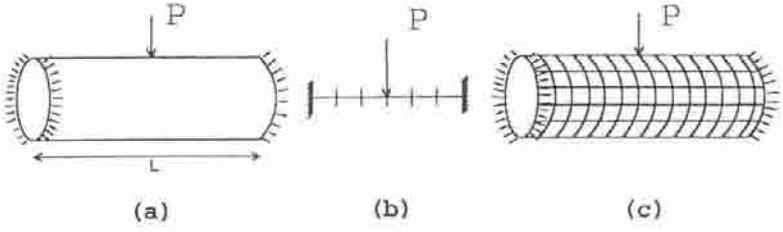

Figure 3.3.1 (a) Cylinder with built-in ends subjected to point load; (b) Beam model; (c) Plate/shell element model.

Certain problems can be dealt with more than one type of analysis. For example, the cylinder under internal pressure shown in Fig. 2.5.1 can be analyzed in at least three different ways. It can be solved as:

a. a plane strain problem. In this case the geometry of the model consists of the circular section obtained by intersecting the cylinder with the YZ plane. By using plane strain elements, we obtain results that are valid for any section of the cylinder that is away from the clamped ends.

b. an axisymmetric problem. The model using axisymmetric elements is shown in Fig. 3.2.6(b) with continuous lines. In this model all nodes on sides AB and CD are constrained, and the pressure p is applied on the side AC. The

deformation and stresses obtained from the analysis of the axisymmetric model are valid for the whole structure. Note that we can analyze the cylinder with this model because the loading is also axisymmetric.

c. a truly three-dimensional structure. The cylinder is modeled using plate elements as shown in Fig. 3.3.1(c). This is the most elaborate and less efficient model; it would be our choice if the load is not axisymmetric.

Combination of Different Types of Elements

In most cases, more than one type of elements must be selected to model the system, e.g., for a bridge structure, plate elements can be used to model the deck, beam elements the piers and girders, solid elements the abutments, and spring elements the foundations. In order to model intersections of pipes and bottom and side tank walls one may use plate elements with a refined mesh at the intersections, see Fig. 3.3.2. However, one may obtain better accuracy for critical areas around the intersections by using plate elements for the pipes or the walls and solid elements for regions at the intersections, see Fig. 3.3.3. For every component modeled with the appropriate element type, the recommendations corresponding to that element type must be followed.

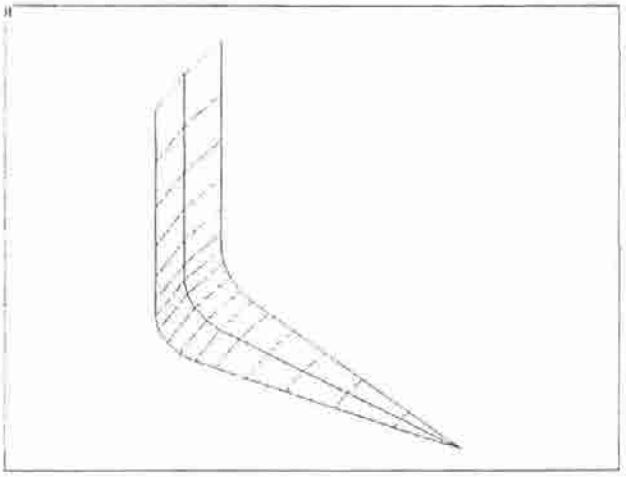

Figure 3.3.2 Plate element model for section of bottom and side tank walls.

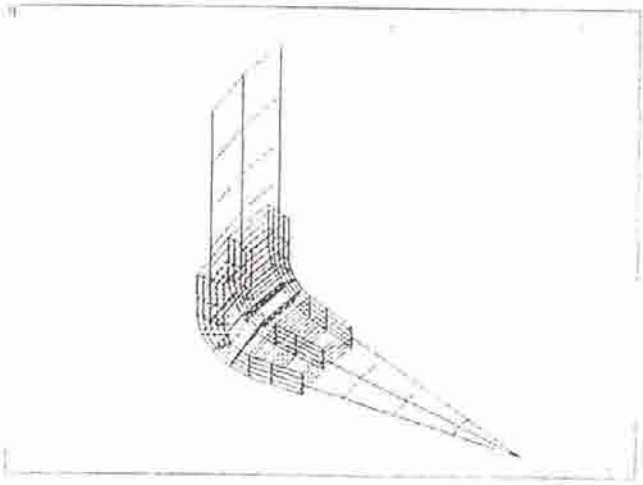

Figure 3.3.3 Combined plate and solid element model for section of bottom and side tank walls.

When combining different types of elements that do not have the same degrees-of-freedom, for example, plate and solid elements or beam and tetrahedral elements, attention should also be paid to the degrees-of-freedom at

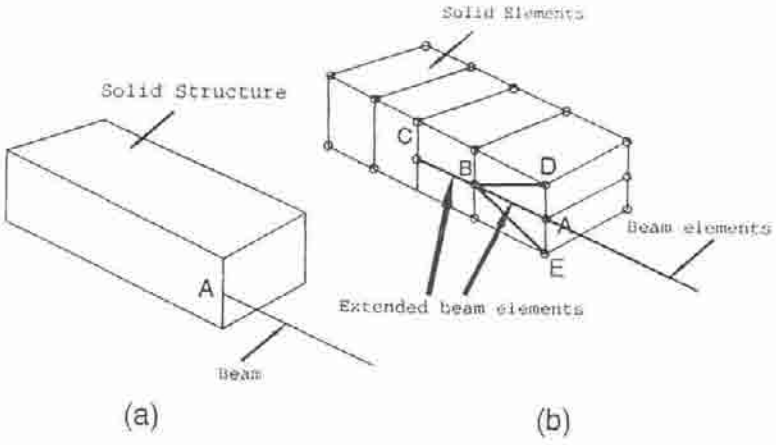

Figure 3.3.4 Modeling of solid-beam intersection.

the common nodes of the different element types. This point is illustrated in Fig. 3.3.4(a). In order to simulate the continuity of stresses and strains at joint A, the beam elements AB, BC, BD, and BE have been added to the model, see Fig. 3.3.4(b).

3.4 MODELING GUIDELINES

Creating the proper model is the most crucial step in finite element analysis. The goal is to develop the most suitable nodal pattern that provides enough elements to obtain accurate results without wasting data interpretation and processing time. Modeling should always be based on conceptual understanding of the physical system and judgment on the anticipated behavior of the structure. Besides understanding the system's behavior, the analyst should make the effort to comprehend the fundamental concepts underlying the pertinent finite element theory. Failure to do so will most likely lead to meaningless results.

The following list of guidelines, accumulated from modeling experience and selected from various sources, can be used as a basis to develop reliable models for static and to a great extent for dynamic analysis. It must be noted that the recommendations may not apply to all cases. No guidelines can substitute for expert use of engineering knowledge and judgement.

General Recommendations

1. Define nodes at or near load points, points of geometric discontinuity, at supports, and at those regions where you require information about stresses or displacements.

2. A finite element mesh should be uniform whenever practical. However, non-uniformity is often required to obtain accurate results in regions of rapid changes in geometry and loading. Only at those parts of the structure where the geometry, stresses, or loading change dramatically should the mesh be refined, see Fig. 3.4.1.

3. Prefer to use quadrilateral, six-sided solid, and hexahedral elements, except where triangular, four-sided solid, and tetrahedral elements are necessary to accommodate irregularities in geometry and loading.

4. When using plate elements to model curved surfaces, maintain a subtended angle of less than 10 degrees, see Fig. 3.2.9.

5. A more refined mesh is required to accurately obtain the stresses than the displacements. If necessary perform a convergence study, i.e., start with a simple model with a relatively small number of elements and progressively move to more refined models. Figure 3.4.2 illustrates the procedure with two plane stress models of a short cantilever beam modeled in the YZ plane. The procedure is also demonstrated through a numerical example in Section 6.6.

Aspect Ratio

The element aspect ratio is defined as the ratio between the element's longest to shortest dimensions. Figure 3.4.3 shows configurations of three two-dimensional element types classified as "good," "poor," and "illegal." Good elements are characterized by an aspect ratio close to unity and angles which are about 90^0. Poor elements should be avoided, since they may lead to inaccurate results. Illegal elements are unacceptable, and should not be used in finite element modeling. When illegal element shapes are inadvertently drawn, they create voids in finite element models. Several postprocessors of commercial finite element programs, such as Algor, allow identification of the voids.

6. If the stress field has similar gradients in all directions, try to maintain an aspect ratio of 1. Elements with high aspect ratio should be avoided, see Fig. 3.4.4(a). However, elements characterized as "poor" in Fig. 3.4.3 can be used in the analysis provided that they model regions of the structure where the stress gradient varies very gradually along the longest dimension of the elements. "Illegal" elements should always be avoided. Because of their geometry and location of the nodes, illegal elements cannot be assigned finite element properties.

7. For regions with small variation of stresses, the aspect ratio could be as high as 40 to 1 and still yield good results. As a general rule, however, keep the element aspect ratios under 10 for *deformation analysis* and under 3 for *stress analysis*, see Fig. 3.4.4(b). It should be noted that in deformation analysis the emphasis is placed on the accurate calculation of the nodal displacements, while in stress analysis we are interested in an accurate calculation of both the displacements and stresses.

8. Rapid changes in element size should be avoided. When the use of different element sizes is necessary, model the transitional parts of the model by changing the dimensions of adjacent elements by less than a factor of two.

Skewed Elements

Skewing is defined as the variation of the element vertex angles from 60° for triangles, and from 90° for quadrilaterals. Notice that two of the "poor" elements in Fig. 3.4.3(b) can be characterized as skewed.

9. For triangular elements avoid acute angles less than 30°, and for quadrilateral elements avoid obtuse angles greater than 120°, see Fig. 3.4.3(b). As a general rule, use of skewed elements is acceptable when we are primarily interested in an accurate calculation of the displacements. If stresses, however, must be calculated accurately, then less skewing should be incorporated to the model. For a discussion on skewed elements see the Algor Software Companion with Verification Examples.

Geometric Discontinuities

10. Holes, cracks, and localized changes in geometry can be characterized as geometric discontinuities. Modeling of geometric discontinuities depends on the interest in accurately calculating deformations and stresses in the vicinity of the discontinuity. If we are not concerned with deformations and stresses near the discontinuity, we can use a *global model* of the structure that has a coarse mesh around the discontinuity. If our goal is to obtain the response near the discontinuity, a *local model* should be used. Contrary to the global model, the local model uses a refined mesh in the vicinity of the discontinuity.

There are two basic approaches to model holes, cracks and notches:

a. The first one involves use of a local model with a refined mesh around the discontinuity. This approach usually requires a detailed mesh that can model the region around the discontinuity in very fine detail.

b. If the stress concentration factor K is known from handbooks or experimental data, a model that calculates the primary stress around the discontinuity can be used. Such a model is less refined than the one required in approach (a).

When there are abrupt geometric changes, one should use a more refined mesh in the region where the changes are located, see Figs. 3.3.2 and 3.4.1. A model similar to the one shown in Fig. 3.3.3 that uses a fine mesh of solid elements to connect plate elements in the region of interest is another modeling alternative.

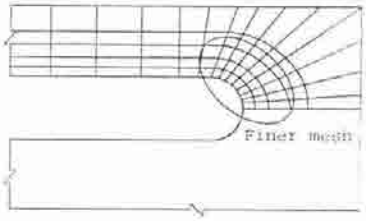

Figure 3.4.1 Plate with stress concentration at slot.

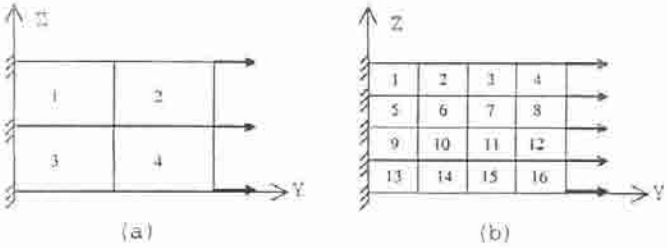

Figure 3.4.2 Four- and sixteen-element models of a cantilever short beam.

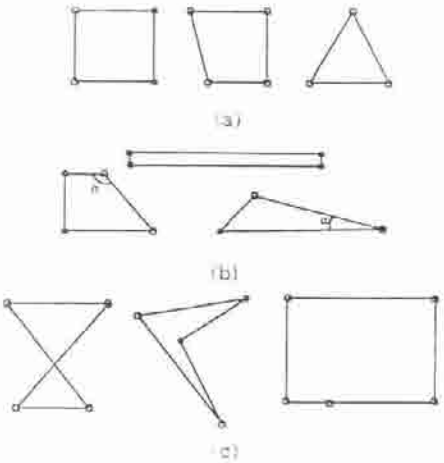

Figure 3.4.3 (a) Good; (b) Poor; (c) Illegal element shapes.

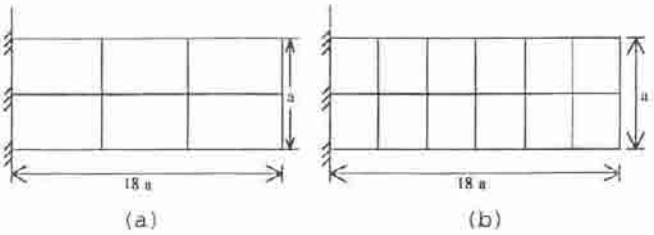

Figure 3.4.4 (a) Very large aspect ratio; (b) Aspect ratio within acceptable range for deformation analysis.

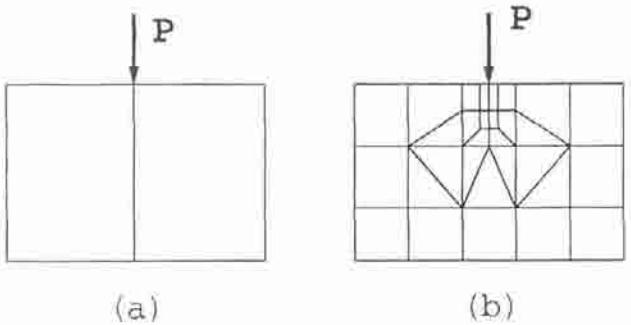

Figure 3.4.5 Mesh in the vicinity of point loads: (a) Coarse; (b) Refined.

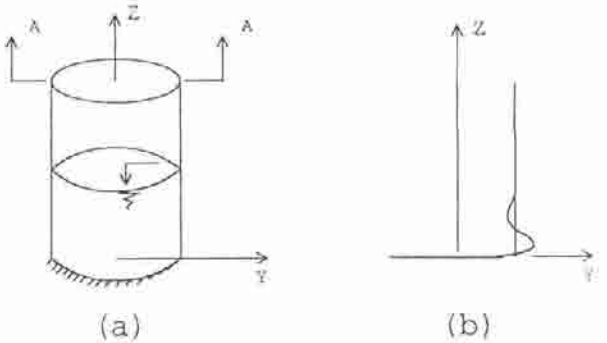

Figure 3.4.6 (a) Liquid filled cylinder; (b) Moment variation and attenuation length in section A-A.

Material Discontinuities

11. Abrupt changes in material properties can be handled with the approaches suggested for "geometric discontinuities," that is, with the aid of global and local models.

12. For isotropic materials, Poisson's ratio must not be very close to 0.5. Also, for a Poisson's ratio close to zero, the structure may lose almost all its stiffness, and as a consequence the results could be erroneous.

13. For anisotropic materials, the theoretical limits of Poisson's ratio and Young's modulus along the specified directions must be verified in order to avoid solution inaccuracies.

Abrupt Changes in Stiffness

14. Avoid models with elements that have stiffness ratios differing more than 10^4. The maximum/minimum stiffness ratios are usually provided in the numerical output of commercial software, e.g., see the "stiffness matrix parameters" in Table 6.5.1.1. Large stiffness variations in a model may lead to singularities and *ill-conditioned* matrices that cannot be numerically handled by the processor. Large stiffness variations could occur in models that contain very small and very large size elements, in models simulating either soft or stiff structural parts, and in models that include stiff or soft elastic boundary and gap elements. A discussion on how to address this issue as well as the proper modeling of stiff and rigid parts of structures is presented in Section 3.7.

Abrupt Changes in Loads

15. A refined mesh should be used in the vicinity of abrupt load changes in order to capture the stress variation near the loads. Such a refined mesh is shown in Fig. 3.4.5. The extent of the region that must be modeled with a fine mesh depends on the attenuation lengths from the points of application of the loads, see Fig. 3.4.6. Various design codes provide attenuation lengths for a variety of engineering applications, e.g., ASME Boiler Code.

3.5 APPLICATION OF BOUNDARY CONDITIONS

Typical boundary conditions include roller, hinge, fixed, and free supports. Rollers are simulated by setting all displacements equal to zero except the ones along the direction that the rollers can move. Typical boundary conditions for a two-dimensional beam are shown in Fig. 3.5.1. Boundary conditions are specified in the global coordinate system.

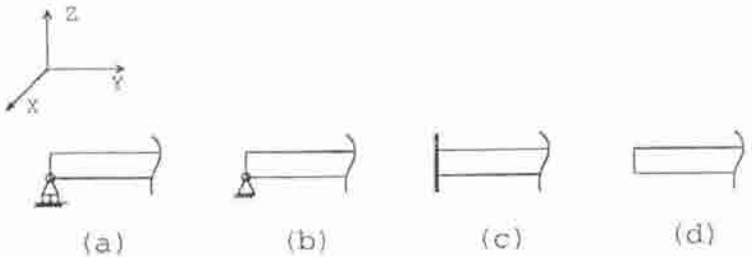

Figure 3.5.1 (a) Roller; (b) Hinge; (c) Fixed; (d) Free boundary conditions.

In problems involving beam elements it may be necessary to include hinged connections. Such connections (*releases*) do not allow the transfer of either moments or shear forces between contiguous beam elements.

Consider the frame shown in Fig. 3.5.2, in which the horizontal nodal displacements at nodes 2, 4 and 6 are equal. This is a common case in frame analysis where the axial stiffness of beams is much greater than the flexural stiffness of the columns. Finite element programs usually allow the user to impose equal displacements at nodes along selected directions through the *master-slave* option, e.g., Algor Software Companion with Verification Examples. Use of the master-slave option could lead to substantial reduction of the degrees-of-freedom, and should be taken advantage of, whenever possible.

Displacements of specified magnitudes and directions as well as skewed boundary conditions can be defined along the boundaries of the structure with the aid of boundary elements. A discussion on how to model skewed boundary conditions is provided in Section 3.8.

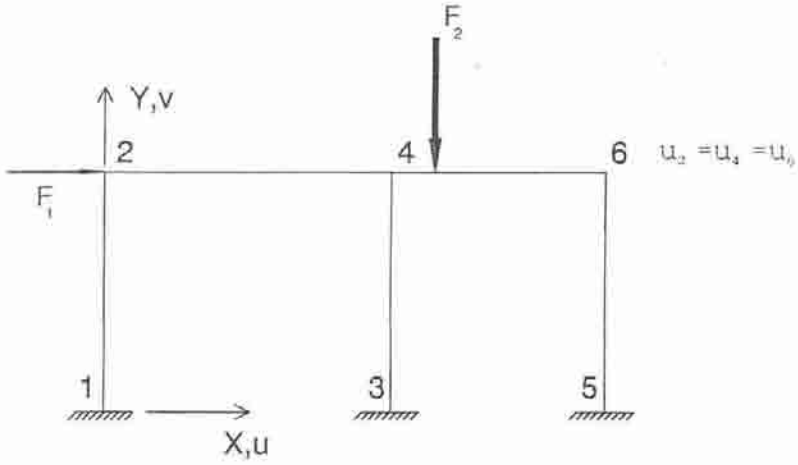

Figure 3.5.2 Master-slave option.

In case of symmetry, boundary conditions must also be specified at the planes of symmetry. Section 3.9 presents pertinent modeling guidelines for static and dynamic analysis.

In any type of analysis except in kinematics and modal analysis, the imposed boundary conditions should not allow rigid body motion. The presence of rigid body motion does not guarantee a unique solution of the problem. Non-uniqueness of solution is reported by the processors of commercial finite element codes.

3.6 APPLICATION OF LOADS

In many statics problems, the contribution of dead loads to the structural deformation could be negligible compared to the effect of live loads. For these cases, setting the mass or the weight density equal to zero not only does not affect the solution but could also save substantial processing time. However, in dynamic analysis either the mass or the weight density of the members must be inputed in order to account for the effects of the inertia forces.

Concentrated loads and reactions cause high stresses in the vicinity of their points of action. Thus, modeling should be very refined close to concentrated loads, if "sufficiently accurate" stresses and deformations are required in the vicinity of the concentrated loads, see Fig. 3.4.5.

In static and dynamic analysis of trusses, only nodal loads can be applied to the truss elements. The nodal loads can be only concentrated forces since no moments can be undertaken by the nodes. In static analysis of beams and frames, commercial finite element programs use *work equivalent* loads to account for the effect of any uniform loads that act on the structure. This implies that the nodal displacements evaluated for frames subjected to either concentrated or uniform loads will be "exact," that is, practically identical to strength of materials solutions, provided that nodes are introduced at the points of application of the concentrated loads and at the ends of the uniform loads, see Fig. 3.6.1. For non-uniform loads, a solution close to the exact can be obtained if additional nodes are introduced between the ends of the non-uniform loads, see Fig. 3.6.2. For dynamic analysis, intermediate nodes are needed for any type of loads acting on a beam, truss, and frame, see Fig. 3.6.3. It should be pointed out that for any other type of elements intermediate nodes are required for both static and dynamic analysis.

For area or volume elements, caution is required in defining the direction of distributed loads, see Fig. 3.6.4. Moments cannot be defined on the nodes of elements that have only translational degrees-of-freedom, such as two- and three-dimensional solid elasticity, tetrahedral, and hexahedral elements. In such cases, if there is a need to define a point moment, M, the moment can be simulated with a couple of opposite forces F acting at a distance d so that M=F*d.

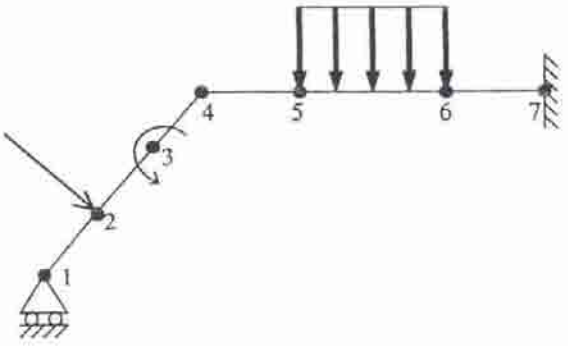

Figure 3.6.1 Model for uniform static load.

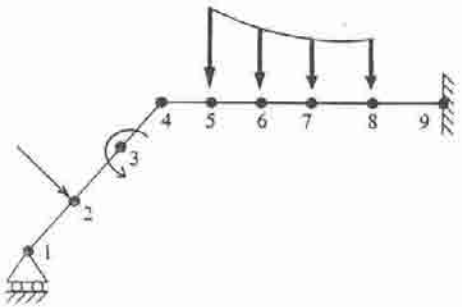

Figure 3.6.2 Model for non-uniform static load.

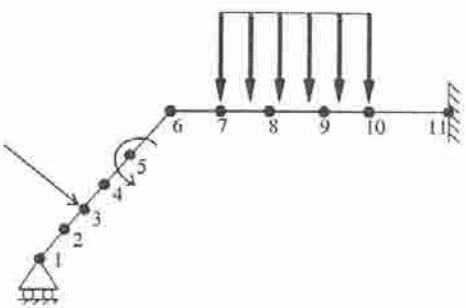

Figure 3.6.3. Model for dynamic analysis of frame.

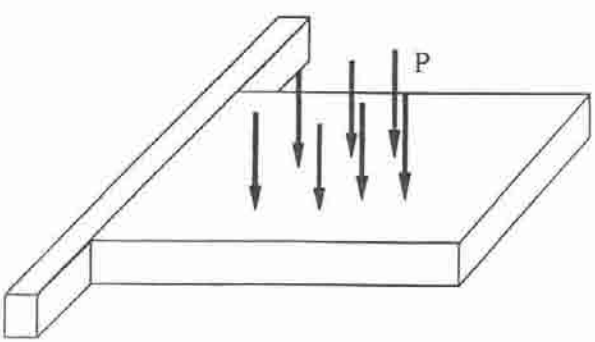

Figure 3.6.4 Plate under uniform load.

3.7 STIFF AND RIGID STRUCTURAL PARTS

Very stiff or almost rigid parts of structures are commonly encountered in practice. For example, such occasions arise when either a portion of the structure or one of its links is much stiffer than the rest of the structure, see Figs. 3.7.1(a) and (b), or when a rather stiff and relatively large object is supported by a flexible structure, see Fig. 3.7.1(c). Detailed modeling of beam-column joints may also require simulation of the joints as rigid, that is assigning an I_R that is much greater than I_1 and I_2, see Fig. 3.7.1(d).

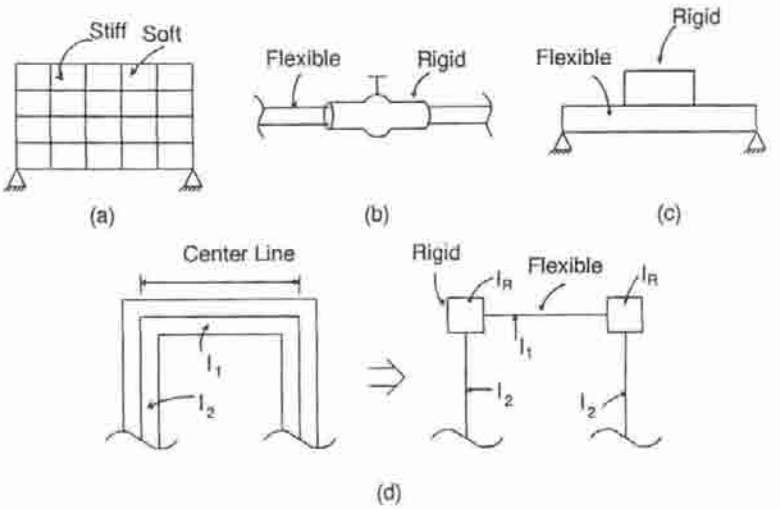

Figure 3.7.1 Modeling of stiff structural parts.

Modeling of very stiff and rigid parts by assigning arbitrarily chosen very high material or cross-sectional properties may lead to either erroneous results or ill-conditioned matrices. A two-step procedure, depicted in Fig. 3.7.2, is recommended in order to avoid such errors. The procedure refers to a structure with elastic properties E_1 that includes stiff and rigid parts with elastic properties E_2. It should be noted that the procedure requires the structure to be analyzed twice.

a. Model the whole structure using a relatively coarse mesh. The whole *structure, including the rigid and stiff parts, should be assigned the elastic* properties E_1. Then, perform a static analysis of the structure to obtain the maximum value of the structure's stiffness matrix, k_{max}, see the "maximum diagonal element" in Table 6.5.1.1.

b. Analyze the structure "as is" but now use a refined mesh and assign to the flexible part its modulus E_1 and to the rigid part a modulus E_2 such that the maximum stiffness coefficient corresponding to an element of the rigid part is in the range of $4k_{max}$ to $5k_{max}$. In plate and frame structures, instead of using E_2, one may chose to assign a large moment of inertia I to the rigid beam elements or a large D to the rigid plate elements, where D is the flexural rigidity of a plate, see eqn. (2.7.2).

The methodology can be easily extended to analyze structures with "soft" parts.

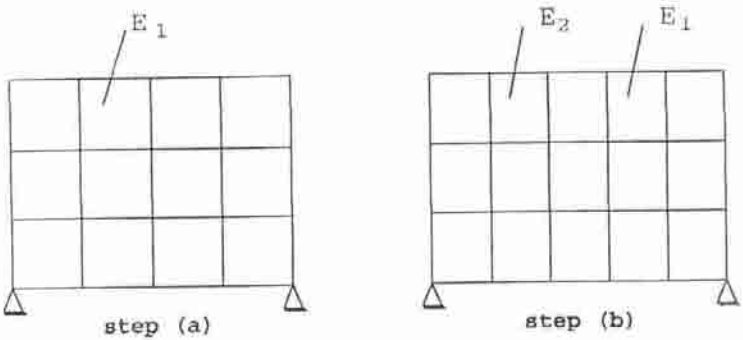

Figure 3.7.2 Two-step approach to model rigid parts.

3.8 SKEWED BOUNDARIES

As mentioned in the Section on "Boundary Elements," modeling and calculation of the reactions at skewed supports can be performed with elastic springs. If the supports are unyielding, a two-step approach can be followed:

a. Model the structure using fixed boundary conditions at the skewed supports. Perform a static analysis to obtain the maximum value of the structure's stiffness matrix, say k_{max}, see "maximum diagonal element" in Table 6.5.1.1.

b. Analyze the structure using translational elastic springs to model the skewed supports. Each spring should be oriented along the direction of the restrained displacement. Also, the stiffness assigned to each spring should be in the range of $4k_{max}$ to $5k_{max}$, in order to warrant sufficient stiffnesses in the springs to restrain the displacements.

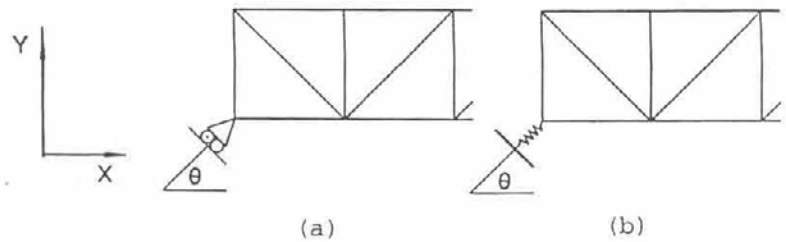

Figure 3.8.1 (a) Skewed boundary condition; (b) Model with elastic boundary element.

3.9 SYMMETRY IN STATIC AND DYNAMIC ANALYSIS

This section presents modeling guidelines for the static and dynamic analysis of symmetric structures. There are several differences between static and dynamic analysis in using symmetry. An objective of this section is to identify these differences and demonstrate the proper use of symmetry through examples. First, we introduce the use of symmetry in static analysis followed by a discussion on dynamic analysis.

Static Analysis

If a structure is symmetric, advantage should be taken of symmetry to reduce the model size and the computational effort. The development of the reduced model requires specification of the appropriate boundary conditions at the planes of symmetry as well as application of the appropriate loads.

In order to characterize a structure as symmetric, one should examine the following three parameters:

a. geometry,
b. material properties, and
c. boundary conditions.

An additional parameter that affects modeling of symmetric structures is the loading.

A structure is symmetric if it has a plane such that the reflection of every point in the structure along this plane results in a similar configuration. The number of planes of symmetry specifies whether the structure has a one-, two-, three-, four-way, or cyclic symmetry.

Figures 3.9.1(a) through (c) show representative examples of one-, two- and four-way symmetric structures subjected to symmetric loads as well as the corresponding reduced models. For the cantilever plate with one axis of symmetry shown in Fig. 3.9.1(a), the boundary conditions for the half model on side AB are T_xR_{yz}. The clamped plate in Fig. 3.9.1(b) is subjected to a uniform load normal to the surface of the plate. For the corresponding quarter model, the boundary conditions on sides BC and DC are T_yR_{xz} and T_xR_{yz}, respectively. In Fig. 3.9.1(c), the boundary conditions on the sides OA and OB are T_yR_{xz} and T_nR_{tz}, respectively. The unit vectors **n** and **t** define the boundary conditions on the side OB. The **t** is parallel to OB and the **n** is normal to the plane defined by **t** and the Z-axis.

When a symmetric structure is subjected to a symmetric load, symmetry restrains displacements normal to the plane of symmetry and rotations in the plane of symmetry. The restrained displacement and rotations are indicated with single and double arrows, respectively, in Fig. 3.9.2(a).

It should be noted that the properties of any members lying on the planes of symmetry should be altered to reflect that only half of these members are a part of the reduced model. Specifically, the members in the reduced model should be assigned one-half their rigidities. For example, the columns on the axis of symmetry of the frame shown in Fig. 3.9.3(a) are assigned one-half their rigidities in the half-model shown in Fig. 3.9.3(b). The magnitude of any load acting at the axis of symmetry should also be reduced, see Figs. 3.9.1 and 3.9.3.

A circular plate with holes subjected to a uniform load normal to its plane and simply supported along its outer boundary is a typical structure with cyclic symmetry, see Fig. 3.9.4. The system can be analyzed by modeling only the section shown in Fig. 3.9.4(b). This section extends between the dashed

lines of Fig. 3.9.4(a). The boundary conditions on the planes of symmetry are defined according to the nodal restrains shown in Fig. 3.9.2(a), that is, along the OA and BC the translation along the Y-axis and the rotations about the X and Z-axis are restrained (T_yR_{xz}), while the boundary conditions on OD are T_uR_{tz}. The boundary conditions on OD can be applied with elastic boundary elements as discussed in Section 3.8.

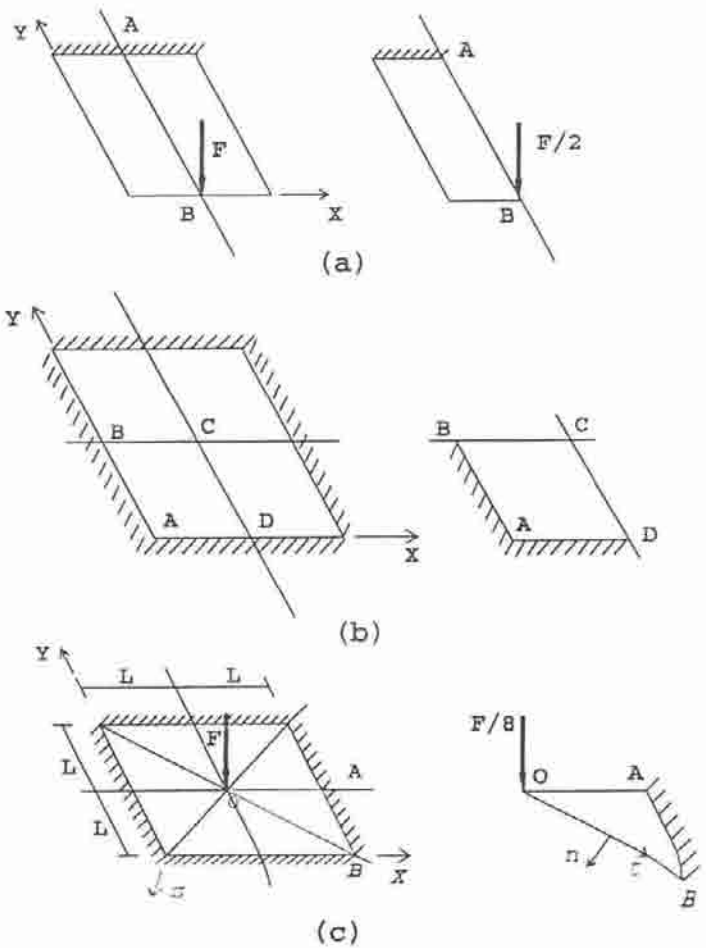

Figure 3.9.1 (a) One-way; (b) Two-way; (c) Four-way symmetry.

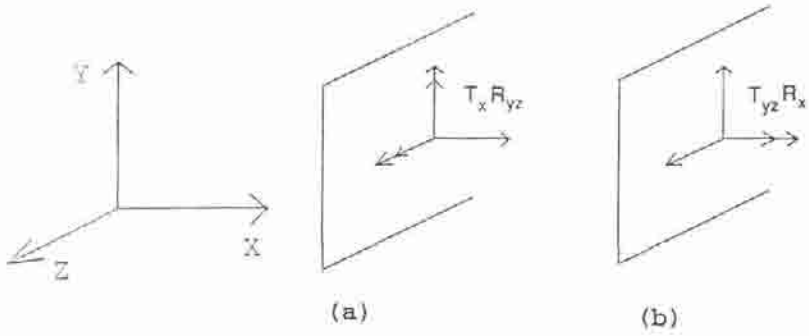

Figure 3.9.2 (a) Restrained degrees-of-freedom for symmetric loads; (b) Restrained degrees-of-freedom for antisymetric loads.

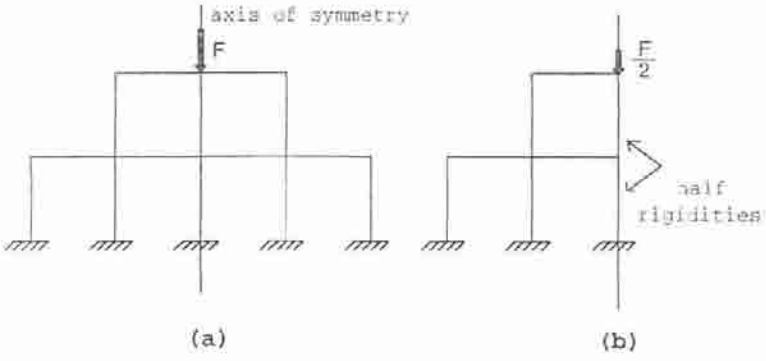

Figure 3.9.3 (a) Symmetric frame; (b) Half structure with reduced rigidities and loads on the axis of symmetry.

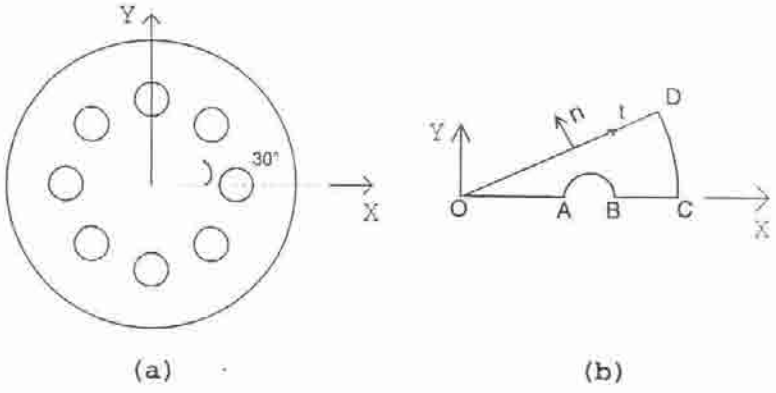

Figure 3.9.4 (a) Plate with cyclic symmetry; (b) Section to model.

When a symmetric structure is subjected to an *antisymmetric* load, the rotation about an axis normal to the plane of symmetry and the displacements in the plane of symmetry should be prevented. The restrained displacements and rotation are indicated with single and double arrows, respectively, in Fig. 3.9.2(b). As shown in Figs. 3.9.5(a) and (b), advantage of symmetry can also be taken when a symmetric structure is subjected to antisymmetric loads. For the frame and the cantilever plate, only half models with appropriate boundary conditions on the axes of symmetry can be analyzed.

If a symmetric structure is subjected to an arbitrary loading, the loads can be decomposed into symmetric and antisymmetric components. In this case, the responses to the symmetric and antisymmetric loading components must be added in order to obtain the response of the part of the structure that was analyzed. By the term *response* we mean the displacements, reactions, internal forces, and stresses. As shown in Figs. 3.9.6, more than two load combinations and analyses may be needed to determine the response of the structure. The response of the frame is obtained by combining the results of the two half models, see Fig. 3.9.6(f). For the plate shown in Fig. 3.9.6(a) with fixed supports on sides AB and DC and with simple supports on the other sides, the results are obtained as a superposition of the four quarter models of Figs. 3.9.6(b) through (e) with the corresponding boundary conditions shown in Figs. 3.9.7 (b) through (e).

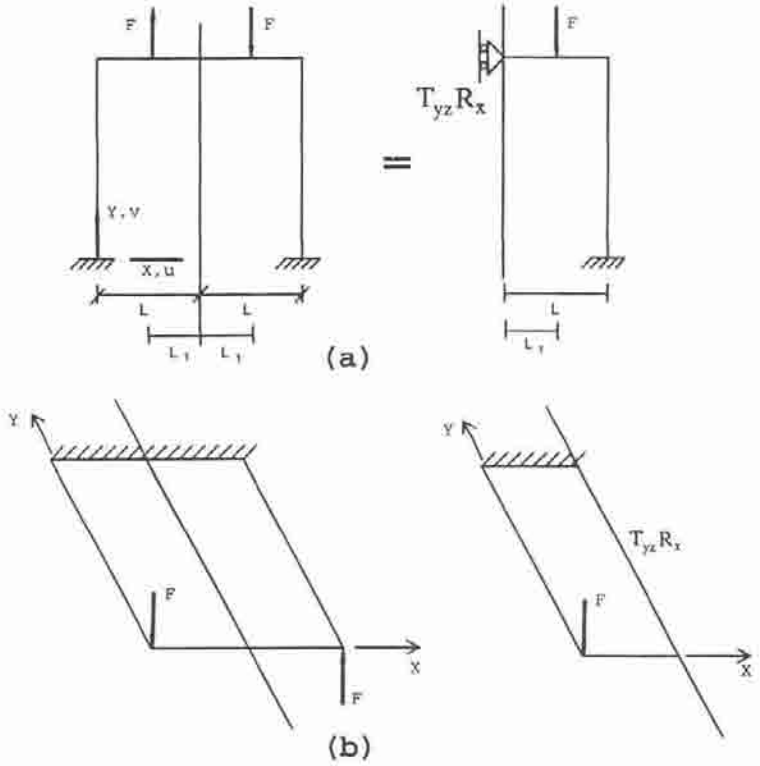

Figure 3.9.5 Symmetric structures subjected to antisymmetric loads: (a) Frame and half model; (b) Plate and half model.

Dynamic Analysis

If a structure fulfills the three requirements of symmetry defined above, then one can also take advantage of symmetry in dynamic analysis. In this section we will illustrate the use of symmetry in modal analysis. Examples on the proper use of symmetry in modal analysis are presented in Chapter 7. Use of symmetry in systems subjected to dynamic loads and analyzed with time history modal superposition is also demonstrated in Chapter 8. Use of symmetry for any other type of dynamic analysis can be deduced from the developments on modal analysis and time history modal superposition.

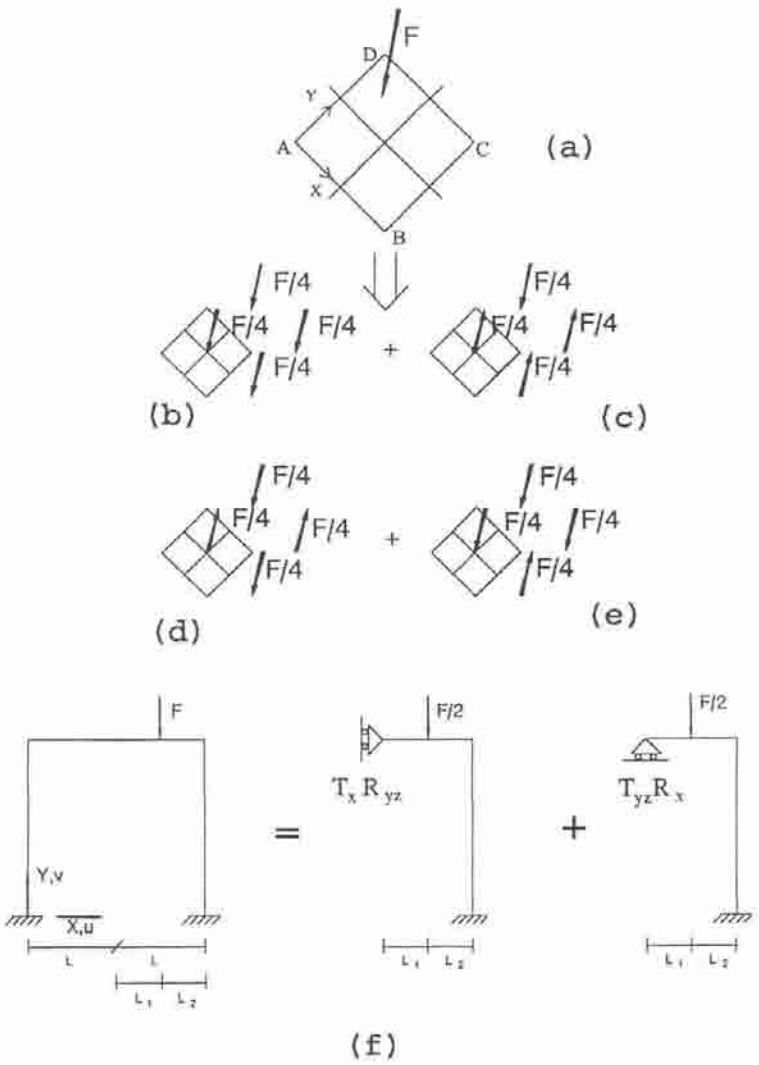

Figure 3.9.6 Symmetric structures subjected to arbitrary loads.

When a system has one or more planes of symmetry, the mode shapes are either symmetric or antisymmetric with respect to the planes of symmetry. Thus, in order to obtain all the mode shapes and natural frequencies we must perform several modal analyses of a part of the structure with proper boundary conditions on the axes of symmetry. The procedure is demonstrated with the modal analysis of a plate that is fully restrained on two opposite sides and simply supported on the others, see Fig. 3.9.7(a). The plate has two planes

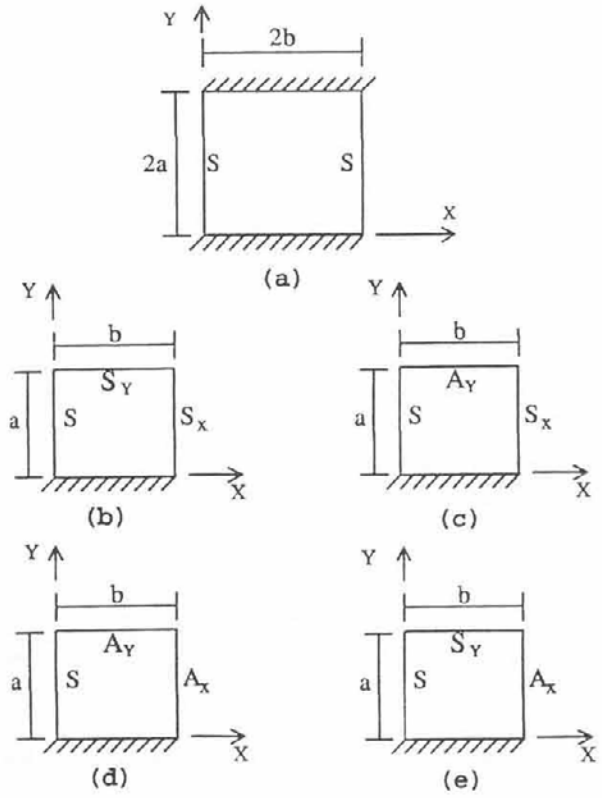

Figure 3.9.7 (a) Symmetric plate; (b)-(e) Quarter models and boundary conditions. Note: The S, S_x, S_y, A_x, A_y indicate the boundary conditions: $T_{xy}R_{xz}$, T_xR_{yz}, T_yR_{xz}, $T_{yz}R_y$, and $T_{xz}R_y$, respectively.

of symmetry, thus we can obtain the natural frequencies and mode shapes of the plate by performing modal analysis on four quarter models. The quarter models and the associated boundary conditions are shown in Figs. 3.9.7(b) through (e). Notice that the only differences between the quarter models are the boundary conditions along the axes of symmetry which are either conditions of symmetry or antisymmetry as specified in Fig. 3.9.2(a) and (b). The corner nodes at the intersection of the planes of symmetry should fulfill the boundary conditions for both intersecting edges, i.e., the conditions at the upper right corner of the plate in Fig. 3.9.7(b) should be $T_{xy}R_{xyz}$. For the modal analysis of this plate with specific geometric and material properties refer to Section 7.3.3.

3.10 RECOMMENDATIONS FOR EVALUATING STRESSES

The evaluation of stresses using finite elements requires a more refined model than the one needed to calculate displacements. This is attributed to the fact that stresses are calculated using derivatives of the displacements. If the finite element mesh is not adequately refined, stresses calculated at adjacent elements may differ substantially. Several programs remove such discrepancies by averaging the stresses in all elements incident to the same node. They also include options to assess the adequacy of the mesh in evaluating stresses, and identifying the parts of the structure that must be modeled with a more refined mesh in order to increase the accuracy of the results.

In evaluating the stress output, the analyst should focus on examining the stress components of interest at *critical locations*. Such critical locations include the boundaries, points of application of concentrated loads, and areas where

a. von Mises, Tresca, or the principal stresses are large,
b. the geometry of the structure changes considerably,
c. the stiffness and/or the mass of the system presents abrupt changes.

Corner nodes are difficult locations to compute stresses. Unfortunately in many cases peak stresses occur at corners, and their magnitudes may govern the design.

As a general rule, the more degrees-of-freedom a node has, the more refined the mesh must be in order to accurately calculate the stresses. For example, the same structure when analyzed under conditions of plane stress, such as a plate loaded parallel to its plane, requires a coarser mesh to calculate the stresses than when analyzed as a plate using plate/shell elements, i.e., loaded normal to its plane. This is attributed to the fact that in the former case plane stress elements with two degrees-of-freedom are used, while in the latter case use is made of plate elements that have five degrees-of-freedom per node. Bending moments in shells and plates can have very high gradients and as a result demand a fine mesh at certain portions of the model, e.g., close to the support of the liquid filled tank shown in Fig. 3.4.6.

Practical Considerations for Modal Analysis

Regardless of what type of analysis is selected to solve a linear dynamics problem, modal analysis should be the first step. Natural frequencies and mode shapes are required for time history modal superposition, frequency response, response spectrum, and random vibration analysis. Even for time history direct integration, that does not require modal analysis, prior calculation of a few modes is recommended, since knowledge of the natural frequencies can help to select the proper time step.

Modal analysis calculates the lowest natural frequencies best, and provides less accuracy in evaluating higher natural frequencies and mode shapes. As a result, there should be sufficient justification to calculate higher modes and include them in subsequent dynamic analysis, e.g., time history modal superposition, response spectrum, and random vibration analysis. As a general rule, the number of modes to request in modal analysis depend on the natural frequencies of the system as related to the frequency content and the spatial variation of the applied loads. The loads with frequencies that are close to the natural frequencies of the system affect the response due to large values of the dynamic magnification factor as elaborated in Section 4.6.2 for a SDOF system. The spatial variation of the loads has an effect on which modes will be mostly excited. For example, a concentrated load acting at the center of a simply supported beam excites only the odd modes, see Section 8.2. The following classification provides additional in-sight in deciding on the proper number of modes that should be obtained with modal analysis.

In general, dynamics problems can be classified into *wave propagation* and *structural dynamics* problems. A typical example of a wave propagation problem is the study of the waves generated in a plate excited by dynamic loads. The evaluation of the plate's response, after the waves have propagated in the whole plate have reflected and refracted on its boundaries, is a structural dynamics problem. The primary difference between wave propagation and structural dynamics is that in wave propagation problems a larger number of mode shapes are excited in the system. In structural dynamics, only the lower and the intermediate modes must usually be determined.

Modal analysis also provides useful insight into the dynamic behavior of a structure. Examination of the mode shapes can help to identify modeling errors such as incorrect boundary conditions and nodal connectivity. Inadvertent omission of restraining nodal displacements or rotations can be identified through rigid body modes of the whole or parts of the model. Additional

remarks and recommendations to perform modal analysis are provided in Section 7.1.

5.5 MODAL ANALYSIS WITH LOAD STIFFENING

The presence of an axial load in a beam affects its flexural response. The degree that compressive static loads influence the flexural behavior leading to *buckling* is greatly dependent upon the way the beam is connected to the rest of the structural system and its slenderness ratio L/r, where r is given by

$$r = \sqrt{\frac{I}{A}} \qquad (5.5.1)$$

with L denoting the unsupported length of the beam, I the moment of inertia, and A the cross-sectional area. Extensive information on buckling caused by static loads is available in the literature, e.g., Chajes.

As for the static case, the presence of axial loads also affects the dynamic flexural response of slender beams. More specifically, compressive loads decrease the flexural stiffness of the beam, while the reverse occurs when the axial loads are tensile. Figures 5.5.1(a) and (b) show the fundamental flexural mode of the same beam under a tensile and a compressive load P, respectively. The fundamental natural frequency of the beam in tension is greater than the fundamental frequency of the beam in compression. In both cases the magnitude of the axial load is kept constant with time.

Both tensile and compressive axial loads can have a significant effect on the flexural dynamic response of slender beams. Similarly, tensile and compressive loads can significantly affect the response of structures that include slender plate and shell components. In practice, however, compressive loads present more interest, since they may lead to failure of the system caused by excessive flexural vibrations. It should be noted that compressive loads primarily affect the fundamental frequency with a lesser effect on higher natural frequencies. The effect is substantial when the axial load is close to the buckling load. Thus, in the presence of large compressive loads, it is prudent to perform a buckling analysis in order to calculate the critical load before a dynamic analysis is carried out. An example in Section 7.7 demonstrates the significance that axial loads play on flexural natural frequencies and mode shapes of beams.

Finally, it should be noted that the flexural response of systems comprised of either beams having small slenderness ratios or thick plates and shells is practically not affected by the presence of axial loads.

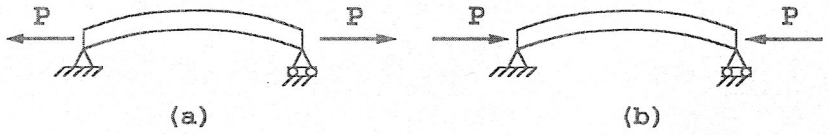

Figure 5.5.1 Fundamental flexural mode of beam subjected to a constant axial load: (a) Tensile; (b) Compressive.

5.6 STURM FREQUENCY CHECK

In Example 5.4.1 the natural frequencies were obtained from the characteristic equation (5.4.7i). Finite element programs employ efficient algorithms to calculate natural frequencies and mode shapes. These algorithms usually include procedures to examine if any natural frequencies were missed in modal analysis. Such a procedure is the Sturm frequency check.

The computational effort to perform the Sturm frequency check is considerable and should be used only when it appears to be necessary. A Sturm frequency check is suggested in the following three cases:

a. when modal analysis is performed to a relatively refined model of the structural system for the first time.
b. when the natural frequencies of the system are closely spaced.
c. when the calculated frequencies appear to be higher than anticipated.

5.7 CUT-OFF FREQUENCY

As we will discuss in more detail in Section 5.11, in many applications only a few low modes contribute significantly to the response. Thus, in many cases calculation of higher modes is unnecessary. The highest frequency in the

analysis is called the cut-off frequency f_c. In finite element programs, specification of f_c is used to terminate the computation of all natural frequencies that are greater than f_c. Only those modes whose frequencies are less than the f_c are then used in subsequent dynamic analysis.

The following suggestions can be of help for an effective use of f_c in modal analysis of large systems:

a. When modal analysis results will be used in a subsequent dynamic analysis of a structure subjected to a seismic excitation, a cut-off frequency of 50 Hz is usually sufficient.

b. When modal analysis results will be used to calculate the response of machinery parts, then a cut-off frequency of as high as 250 Hz may be selected.

5.8 RIGID BODY MODES

The natural frequencies of a system can either be positive or zero. The frequencies that are zero correspond to rigid body modes, i.e., modes in which none of the elastic elements is deformed. A representative example is an airplane body and wings modeled as a flexible beam with three lumped masses. The mass of the fuselage is denoted as m_f, and m_w represents the mass of one wing. The first three modes of the three lumped mass model are shown in Fig. 5.8.1. Notice that the first two modes are rigid body modes with frequencies equal to zero.

Three-dimensional models that are deliberately unsupported exhibit up to six rigid body modes. Any additional zero natural frequencies either correspond to parts of the model that respond as mechanisms or have been introduced inadvertently by not specifying sufficient boundary conditions to restrain rigid body displacements. In the latter case, errors in providing inappropriate connectivity between structural parts can be detected by examining the corresponding mode shapes. Rigid body modes can also be present in structures supported on soft springs, see Fig. 5.8.2.

An example of a structure with rigid body modes is presented in Chapter 7.

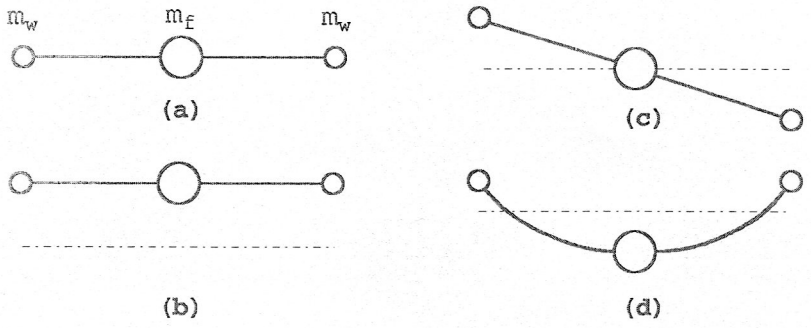

Figure 5.8.1 (a) Simple airplane model; Three lowest modes: (b) Fundamental mode (rigid body translation); (c) Second mode (rigid body rotation); (d) Third mode (flexural deformation).

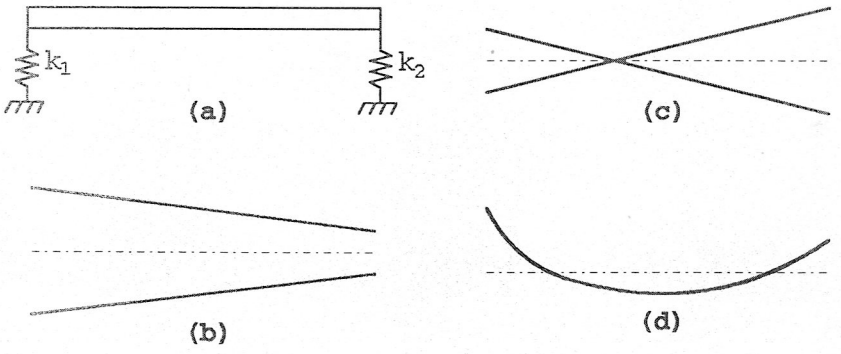

Figure 5.8.2 (a) Beam on unequal "soft" springs; Three lowest modes: (b) First mode (rigid body translation); (c) Second mode (rigid body rotation); (d) Third mode (flexural deformation).

5.9 HOW TO INCORPORATE DAMPING IN FEM

In modal analysis damping is neglected because it is usually small and practically does not affect the calculation of natural frequencies and mode shapes, see Section 4.4. Also, damping can be neglected in systems with viscous damping ratios less than 0.02, provided the excitation is harmonic with frequency ratios that are not in the range of 0.5 to 1.45, see Section 4.6.2. Nevertheless, damping is one of the primary concerns in dynamic analysis and is accounted for with the damping matrix [C]. In general, the matrix [C] cannot be constructed from element damping matrices, as is the case for the mass and stiffness matrices. The most effective computational procedure to account for damping is to express [C] either as an algebraic combination of the mass and stiffness matrices of the system or in terms of damping ratios and natural circular frequencies. These procedures are elucidated in Sections 5.10 and 5.12.

In response spectrum analysis, damping is incorporated into the response spectra. As presented in Section 4.9, one of the parameters in selecting the proper spectra is the amount of damping anticipated in the system. In time history modal superposition, frequency response, and random vibration, damping is usually expressed in the form of equivalent viscous, while Rayleigh damping is usually used in time history direct integration. Particular details about consideration of damping for each type of dynamic analysis are given in Chapters 5, 8, and 9.

Regarding the significance that accurate consideration of damping has on the solution accuracy, the rule is that the longer the duration of the dynamic loads the greater the effect of damping on the system's response. Thus, for blast or shock loadings damping plays very little role on the response and as a result it is usually neglected. For seismic loads, damping can significantly reduce the response and therefore it must be included in the analysis. Damping should be accurately specified for the modes that have the largest contribution to the response. These modes can be identified by comparing the natural frequencies of the structure to the frequencies of the harmonic terms of the Fourier analysis of the dynamic loads. Thus, if a Fourier series expansion or a Fourier transform of the loads shows that only frequencies up to f_s are included in the loading, the modal analysis should accurately calculate modes up to $f_c = 4f_s$. If the frequency content of the load is not determined through a Fourier transform or a series expansion, then an estimate of the f_c can be obtained from the suggestions given in Section 5.7.

Clearly, eqn. (5.10.2m) indicates that the response is obtained as a superposition of the modal responses. *Modal response* is defined as the product of the the i-th mode shape with the corresponding $z_i(t)$. If we perform the multiplications indicated in eqn. (5.10.2m), the solution can also be written in the form

$$y_1(t) = 0.102 z_1(t) - 0.102 z_2(t)$$

$$y_2(t) = 0.072 z_1(t) + 0.072 z_2(t) \qquad (5.10.2n)$$

For an N-degree-of-freedom system the counterpart of eqn. (5.10.2m) is given by

$$\begin{Bmatrix} y_1 \\ y_2 \\ . \\ . \\ y_N \end{Bmatrix} = \begin{Bmatrix} \phi_{11} \\ \phi_{21} \\ . \\ . \\ \phi_{N1} \end{Bmatrix} z_1(t) + \begin{Bmatrix} \phi_{12} \\ \phi_{22} \\ . \\ . \\ \phi_{N2} \end{Bmatrix} z_2(t) + \ldots + \begin{Bmatrix} \phi_{1N} \\ \phi_{2N} \\ . \\ . \\ \phi_{NN} \end{Bmatrix} z_N(t) \qquad (5.10.2o)$$

5.11 PRACTICAL CONSIDERATIONS FOR TIME HISTORY MODAL SUPERPOSITION

Finite element programs use efficient algorithms to perform the "uncoupling of the equations of motion" and the "superposition of the independent solutions" described in Example 5.10.1. The response is not provided in a closed form but rather at user defined time increments (time steps) of duration Dt. Such efficient algorithms include the Wilson's θ and the Newmark's β methods. Both methods are *unconditionally stable*, which implies that they provide the response of the system regardless of the selected time step Dt. However, selection of an arbitrary value for Dt does not warrant that the obtained response is accurate. The selection of Dt is very critical, since it affects

both the computational efficiency and the solution accuracy. A large value for Dt will provide inaccurate results, while a very small Dt will unnecessarily increase the computational time and cost, without a justifiable improvement in solution accuracy. A recommendation for the selection of a proper Dt is given in the following.

A five-step procedure can be used to obtain the response of a system subjected to dynamic loads with time history modal superposition. Even though the first step can be omitted, its execution greatly expedites the solution, since it can be of help in developing the proper model and determining the appropriate natural frequencies and mode shapes through modal analysis.

1. Determine the frequency content of the dynamic loads. For a periodic loading this can be easily accomplished with a Fourier series expansion. When the load is transient the frequency content can be obtained with a Fast Fourier Transform, see Section 4.11.1. For several common types of dynamic loads, approximate upper limits of the frequency content have been presented in Section 5.7.

2. Perform modal analysis on a finite element model that can accurately calculate natural frequencies up to f_c. For a discussion on the critical frequency f_c and selection of the number of modes to request in modal analysis refer to Sections 5.4, 5.7, and 5.9.

3. Select a time step Dt that minimizes the computational inaccuracies inherent in the integration scheme used in time history modal superposition. As a general rule, a time step $Dt < 0.05T_c$, where $T_c = 1/f_c$, provides accurate results. In most cases, however, a $Dt < 0.1T_c$ is sufficient.

4. Select the appropriate values for the damping ratio ξ according to the recommendations given in Section 4.12.

5. Define the duration that the response should be evaluated. In selecting the duration of the analysis, we must consider the type and duration of the load as well as the amount of damping in the system. For lightly damped systems ($0.0 < \xi < 0.05$), the analysis could be performed for about $10\,T_1$, where T_1 is the fundamental period of the system. For more heavily damped systems, the analysis could be performed for about $5T_1$ to $8T_1$, see Fig. 4.6.3. For impact loads we ususally ignore damping and calculate the response for a duration slightly greater than the duration of the load. A discussion on the effect of loading duration on the response is presented in Section 4.6.3. The discussion in Section 4.6.3 refers to harmonic loads. The conclusions, however, apply as a conservative extension to general excitations.

Examples in Chapter 8 illustrate various aspects of the five-step procedure.

5.12 TIME HISTORY DIRECT INTEGRATION

An alternative method to obtain the response of a system to dynamic loads is the time history direct integration. This method uses time step-by-step integration algorithms to solve the system of the coupled equations of motion, i.e., eqns. (5.2.1). Such algorithms include Wilson's θ and Newmark's β methods that calculate the response at specified time intervals. As mentioned in the previous section, both methods are unconditionally stable, which means that whatever the selection of the time interval Dt, the solution scheme will provide an answer. However, this answer may not necessarily be correct.

As in modal superposition, an accurate solution is obtained with a $Dt < 0.05T_c$. For most applications, however, we can obtain a sufficiently accurate solution with a Dt that satisfies the relationship

$$Dt \leq 0.1\,T_c \qquad (5.12.1)$$

where T_c is the smallest natural period that should be accounted for in the analysis. Suggestions for selecting the proper f_c and the corresponding T_c are presented in Sections 5.4, 5.7, and 5.9. A larger value of Dt could still provide a sufficiently accurate solution, but contribution of modes with natural frequencies greater than T_c will be *filtered*, i.e., their contribution to the response will be significantly decreased.

One of the main differences between time history modal superposition and direct integration is the way damping is treated by the two methods. In principle, direct integration does not require the damping in the system to be viscous; the damping matrix does not have to satisfy any orthogonality properties that allow decoupling of eqn. (5.2.1). In practice, however, the damping matrix is constructed by assigning selected values to the damping ratios ξ_i. In direct integration *Rayleigh damping* is used. For Rayleigh damping, the damping matrix is given by

$$[C] = \alpha [M] + \beta [K] \qquad (5.12.2)$$

The constants α and β are calculated from the system of equations

$$\alpha + \beta \omega_i^2 = 2\omega_i \xi_i \qquad (5.12.3)$$

where ω_i is obtained through modal analysis, and ξ_i are damping ratios specified by the analyst. From eqn. (5.12.3), we can easily prove that if we assign two damping ratios ξ_i and ξ_j corresponding to the natural circular frequencies ω_i and ω_j, the α and β can be determined from

$$\alpha = \frac{2\omega_i \omega_j}{\omega_j^2 - \omega_i^2}(\omega_j \xi_i - \omega_i \xi_j)$$

$$\beta = \frac{2}{\omega_j^2 - \omega_i^2}(\omega_j \xi_j - \omega_i \xi_i) \qquad (5.12.4)$$

Rayleigh damping, being a linear combination of the mass and stiffness matrices, fulfills the orthogonality conditions (5.4.7s) and (5.4.7t), which in turn imply that Rayleigh damping allows decoupling of the equations of motion. It is important to note that if $[C]=\alpha[M]$, ($\beta=0$), the higher modes of the structure will be assigned very little damping, while for $[C]=\beta[K]$, ($\alpha=0$), the higher modes will be heavily damped. Thus, by giving appropriate values to α and β, we can filter or retain the effect of the higher modes.

A numerical example demonstrating the procedure to calculate α and β is also presented in Section 8.4.1. As mentioned earlier, time history direct integration does not require prior modal analysis. Nevertheless, the fact that Rayleigh damping is expressed in terms of damping ratios and natural circular frequencies implies that, in damped systems, modal analysis must precede time history direct integration, except if the natural frequencies and the damping are already known.

5.13 TIME HISTORY: MODAL SUPERPOSITION VERSUS DIRECT INTEGRATION

In dynamic analysis using time history, the time at which each one of the forces or the ground excitation start to act on the system must be specified. Since the part of the response caused by the initial conditions is usually diminished very rapidly due to damping, common practice sets the initial

conditions, i.e., displacements, velocities, and accelerations equal to zero.

From the discussion in Sections 5.10 through 5.12, it should be clear now that there are more similarities and fewer differences between modal superposition and direct integration. For the same time step Dt, direct integration provides identical results to modal superposition for which all significant natural frequencies and mode shapes are included in the analysis. The remaining part of this section emphasizes the differences between the two methods.

The main advantage of direct integration over modal superposition is that direct integration can be used for both linear and nonlinear dynamics problems. Direct integration should be preferred when many modes must be included and the response is required for a short duration of time, such as in shock vibration problems. Direct integration provides more choices in accounting for damping. In principle, the damping matrix [C] does not have to satisfy the orthogonality conditions in direct integration analysis. However, use of Rayleigh damping implies modal orthogonality.

Contrary to direct integration, modal analysis must precede modal superposition. This is an advantage of modal superposition, since modal analysis allows detection of modeling errors as well as selection of the proper Dt. A second advantage of modal superposition is that, once a modal analysis of the system is performed, several modal superposition analyses can be performed for the same system subjected to different loads with a relatively small additional computational effort. Modal superposition is preferred over direct integration when the duration of the loads is long and a relatively small number of modes contributes to the response.

5.14 FREQUENCY RESPONSE ANALYSIS

If the externally applied loads or the ground excitation are harmonic, the preferred method of analysis is usually frequency response analysis. As demonstrated in Example 5.14.1 and later in Chapter 9, frequency response analysis provides the steady-state response. The contribution of the transient part to the system's response cannot be captured with frequency response analysis. Thus, strictly speaking, the results obtained with frequency response analysis are not conservative for design. If we want to determine both the transient and the steady-state part of the response, then we should choose either time history modal superposition or direct integration. When, however, the duration of the transient part of the response is either very short or not of