❖ Mesh Generation

- The network of elements and nodes that discretize a region is referred to as a mesh.
- The mesh density increases as more elements are placed within a given region.
- Mesh refinement is when the mesh is modified from one analysis of a model to the next analysis to yield improved results.
- Results generally improve when the mesh density is increased in areas of high stress gradients and/or when geometric transition zones are meshed smoothly.
- There are three basic ways to generate an element mesh, manually, semiautomatically, or fully automated.

1- **Manual mesh generation**. This is how the element mesh was created in the early days of the finite-element method. This is a very labor intensive method of creating the mesh.

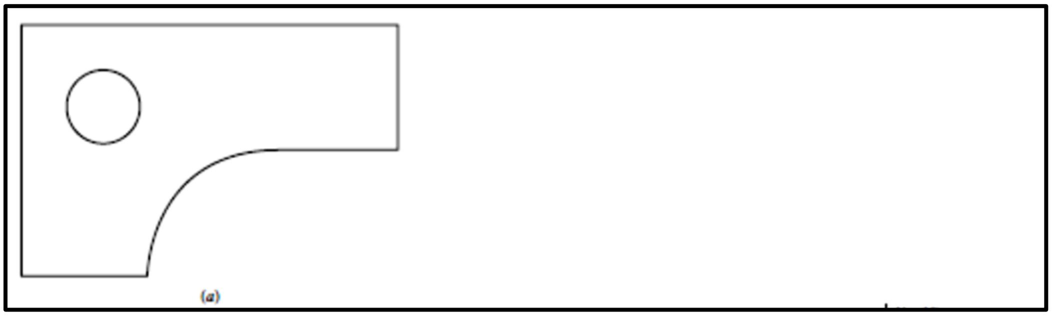
2- **Semiautomatic mesh generation.** Over the years, computer algorithms have been developed that enable the modeler to automatically mesh regions of the structure that he or she has divided up, using well-defined boundaries.

3- **Fully automated mesh generation.** Many software vendors have concentrated their efforts on developing fully automatic mesh generation, and in some instances, automatic *self-adaptive* mesh refinement. The obvious goal is to significantly reduce the modeler's preprocessing time and effort to arrive at a final well-constructed FEA mesh. Various schemes are available to discretize the region with *one element type*.

- For plane elastic problems the boundary is defined by a series of internal and external geometric lines and the element type to be automeshed would be the plane elastic element.
- For thin-walled structures, the geometry would be defined by three-dimensional surface representations and the automeshed element type would be the three-dimensional plate element.
- For solid structures, the boundary could be constructed by using *constructive solid geometry* (*CSG*) or *boundary representation* (*B-rep*) techniques.
- The finite-element types for automeshing would be the brick and/or tetrahedron.

▪ Returning to the thin-plate model of Fig. 19–2, the boundaries of the structure are constructed as shown in Fig. 19–6*a*.
▪ The boundaries were then automeshed as shown in Fig. 19–6*b*, where 294 elements and 344 nodes were generated. Note the uniformity of the element generation at the boundaries.

- The finite-element solver then generated the deflections and von Mises stresses shown in Fig. 19–6*c*. The maximum von Mises stress at the location shown is 4110.4 psi.
- The model was then automeshed with an increased mesh density as shown in Fig. 19–6*d*, where the model has 1008 elements and 1096 nodes.
- The results are shown in Fig. 19–6*e* where the maximum von Mises stress is found to be 4184.9 psi, which is only 1.8 percent higher. In all likelihood, the solution has nearly converged. *Note:* The stress contours of Figs. 19–6*c* and *e* are better visualized in color.



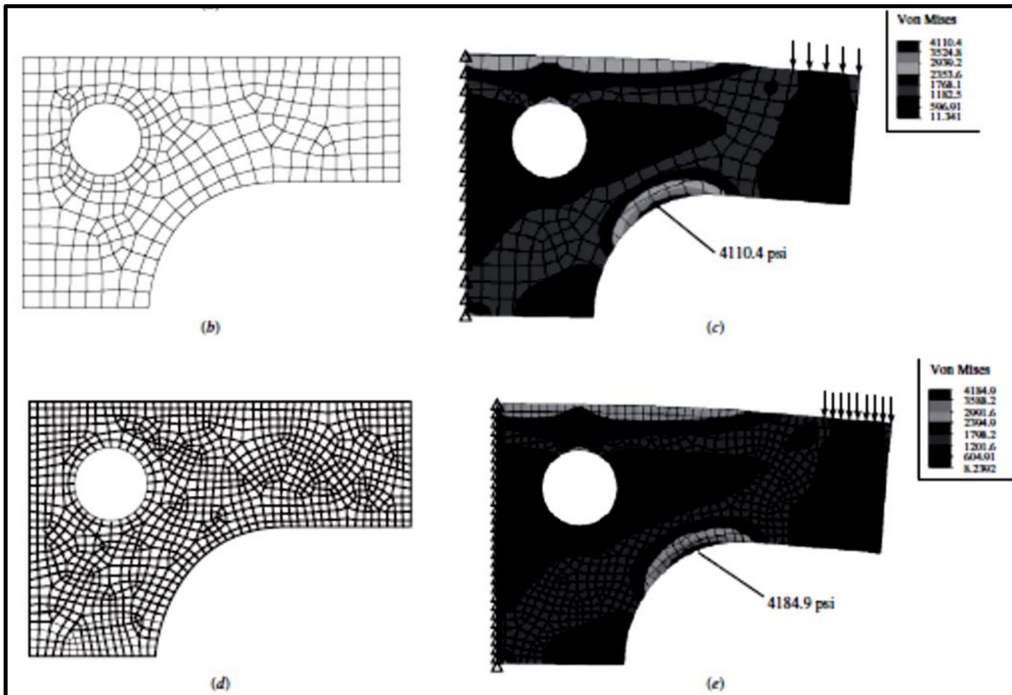


Figure 19–6

Automatic meshing the thin-plate model of Fig. 19–2. (*a*) Model boundaries; (*b*) automesh with 294 elements and 344 nodes; (*c*) deflected (exaggerated scale) with stress contours; (*d*) automesh with 1008 elements and 1096 nodes, (*e*) deflected (exaggerated scale) with stress contours.

❖ Load Application

- There are two basic forms of specifying loads on a structure, nodal and element loading. However, element loads are eventually applied to the nodes by using

equivalent nodal loads.

- The net force and/or moment can be applied to a single node, provided the element supports the dof associated with the force and/or moment at the node.
- Concentrated moments can be applied to the nodes of beam and most plate elements. However, concentrated moments cannot be applied to truss, two-dimensional plane elastic, axisymmetric, or brick elements. They do not support rotational degrees of freedom.
- A pure moment can be applied to these elements only by using forces in the form of a couple. From the mechanics of statics, a couple can be generated by using two or more forces acting in a plane where the net force from the forces is zero. The net moment from the forces is a vector perpendicular to the plane and is the summation of the moments from the forces taken about any common point.
- Element loads include static loads due to gravity (weight), thermal effects, surface loads such as uniform and hydrostatic pressure, and dynamic loads due to constantacceleration and steady-state rotation (centrifugal acceleration).
- For thermal loading, the thermal expansion coefficient α must be given for each material, as well as the initial temperature of the structure, and the final nodal temperatures.
- Surface loading can generally be applied to most elements. For example, uniform or linear transverse line loads (force/length) can be specified on beams. Uniform and linear pressure can normally be applied on the edges of two-dimensional plane and axisymmetric elements. Lateral pressure can be applied on plate elements, and pressure can be applied on the surface of solid brick elements.

❖ Boundary Conditions

The simulation of boundary conditions and other forms of constraint is probably the most difficult part of the accurate modeling of a structure for a finite-element analysis. For example, we have modeled shafts with bearings as being simply supported. It is more likely that the support is something between simply supported and fixed, and we could analyze both constraints to establish the limits. However, by assuming simply supported, the results of the solution are conservative for stress and deflections. That is, the solution would predict stresses and deflections larger than the actual.

For another example, consider beam 16 in Table A–9. The horizontal beam is uniformly loaded and is fixed at both ends. Although not explicitly stated, tables such as these

assume that the beams are not restrained in the horizontal direction.

Multipoint constraint equations are quite often used to model boundary conditions or rigid connections between elastic members. When used in the latter form, the equations are acting as elements and are thus referred to as *rigid elements*. Rigid elements can rotate or translate only rigidly.

Boundary elements are used to force specific nonzero displacements on a structure. Boundary elements can also be useful in modeling boundary conditions that are a skew from the global coordinate system.

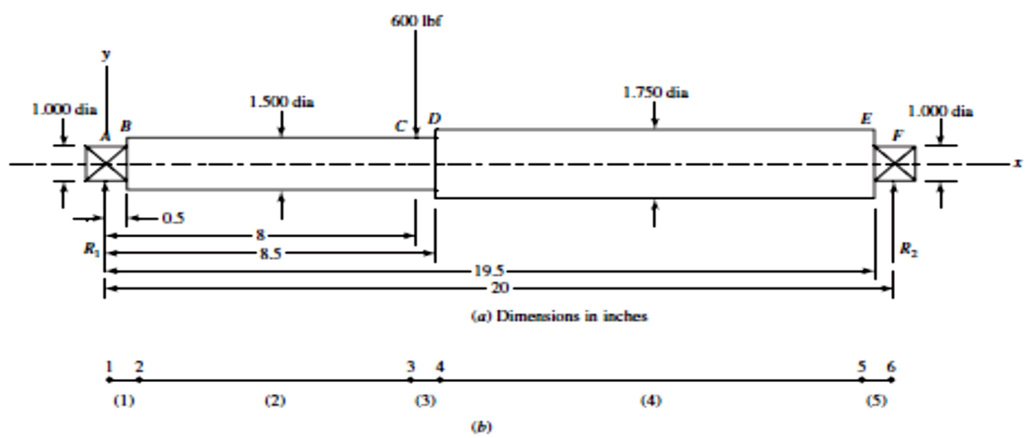
❖ Modeling Techniques

With today's CAD packages and automatic mesh generators, it is an easy task to create a solid model and mesh the volume with finite elements. With today's computing speeds and with gobs of computer memory, it is very easy to create a model with extremely large numbers of elements and nodes.

The complex model may not even provide an accurate solution, whereas a simpler one will. What is important is what solution the analyst is looking for: deflections, stresses, or both?

For example, consider the steel step-shaft in Fig. 19–7*a*. Let the fillets at the steps have a radius of 0.02 in. If only deflections and slopes were sought at the steps, a highly meshed solid model would not yield much more than the simple five-element beam model, shown in Fig. 19–7*b*, would. The displacement results for the FEA model are shown in Fig. 19–7*c*.

The FE model of Fig. 19–7*b* is not capable of providing the stress at the fillet of the step at *D*. Here, a full-blown solid model would have to be developed and meshed, using



(a) Dimensions in inches

(b)

Displacements/rotations (degrees) of nodes

NODE no.	x translation	y translation	z translation	θ_x rotation (deg)	θ_y rotation (deg)	θ_z rotation (deg)
1	0.0000 e + 00	0.0000 e + 00	0.0000 e + 00	0.0000 e + 00	0.0000 e + 00	−9.7930 e − 02
2	0.0000 e + 00	−8.4951 e − 04	0.0000 e + 00	0.0000 e + 00	0.0000 e + 00	−9.6179 e − 02
3	0.0000 e + 00	−9.3649 e − 03	0.0000 e + 00	0.0000 e + 00	0.0000 e + 00	−7.9874 e − 03
4	0.0000 e + 00	−9.3870 e − 03	0.0000 e + 00	0.0000 e + 00	0.0000 e + 00	2.8492 e − 03
5	0.0000 e + 00	−6.0507 e − 04	0.0000 e + 00	0.0000 e + 00	0.0000 e + 00	6.8558 e − 02
6	0.0000 e + 00	0.0000 e + 00	0.0000 e + 00	0.0000 e + 00	0.0000 e + 00	6.9725 e − 02

(C)

Figure 19–7

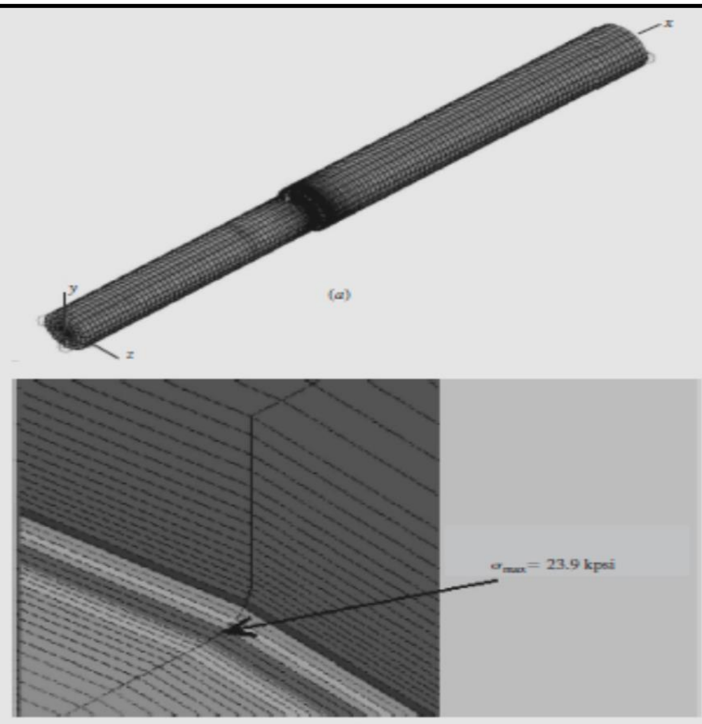
(a) Steel step shaft of Ex. 4–7; (b) finite-element model using five beam elements; (c) displacement results for FEA model.

solid elements with a high mesh density at the fillet as shown in Fig. 19–8*a*. Here, the steps at the bearing supports are not modeled, as we are concerned only with the stress concentration at x = 8.5 in. The brick and tetrahedron elements do not support rotational degrees of freedom. To model the simply supported boundary condition at the left end, nodes along the z axis were constrained from translating in the x and y directions.

The maximum tensile stress at the fillet at the beam bottom is found to be σmax = 23.9 *kpsi*. Performing an analytical check at the step yields D/d = 1.75/1.5 = 1.167, and r/d = 0.02/1.5 = 0.0133. Figure A–15–9 is not very accurate for these values.

Figure 19–8

(a) Solid model of the step-shaft of Ex. 4–7 using 56 384 brick and tetrahedron elements; (b) view of stress contours at step, rotated 180° about x axis, showing maximum tension.



Resorting to another source,13 the stress concentration factor is found to be Kt = 3.00.

The reaction at the right support is R_F = (8/20)600 = 240 *lbf*. The bending moment at the start of the fillet is M = 240(11.52) = 2 765 *lbf* · in = 2.765 kip · in. The analytical prediction of the maximum stress is thus

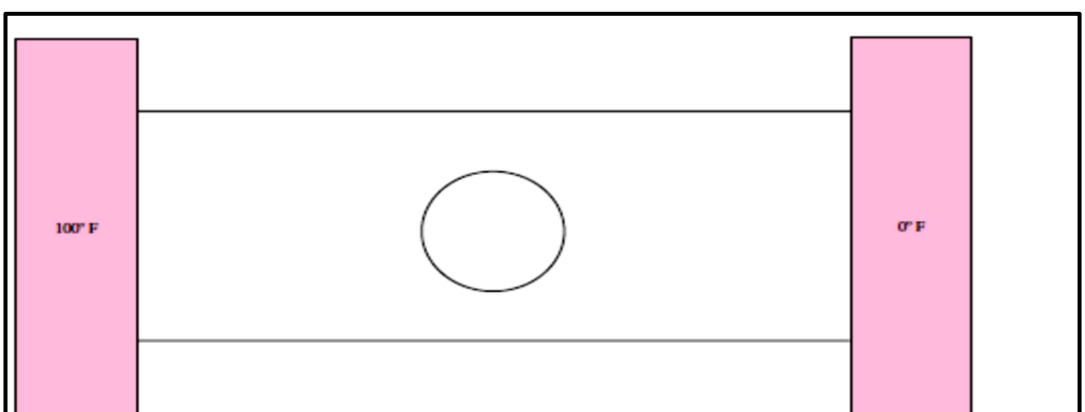
$$\sigma_{max} = K_t\left(\frac{32M}{\pi d^3}\right) = 3.00\left[\frac{32(2.765)}{\pi(1.5^3)}\right] = 25.03 \text{ kpsi}$$

The finite-element model is 4.5 percent lower. If more elements were used in the fillet region, the results would undoubtedly be closer. However, the results are within engineering acceptability.

❖ Thermal Stresses

A heat transfer analysis can be performed on a structural component including the effects of heat conduction, convection, and/or radiation. After the heat transfer analysis is completed, the same model can be used to determine the resulting thermal stresses.

For sake of a simple illustration, we will model a 10 in × 4 in, 0.25-in-thick steel plate with a centered 1.0-in-diameter hole. The plate is supported as shown in Fig. 19–9*a*, and the temperatures of the ends are maintained at temperatures of 100˚F and 0˚F. Other than at the walls, all surfaces are thermally insulated. Before placing the plate between the walls, the initial temperature of the plate was 0˚F. The thermal coefficient of expansion for steel is α_s = 6.5 × 10−6 ˚F −1. The plate was meshed with 1312 two-dimensional elements, with the mesh refined along the border of the hole.



(a)

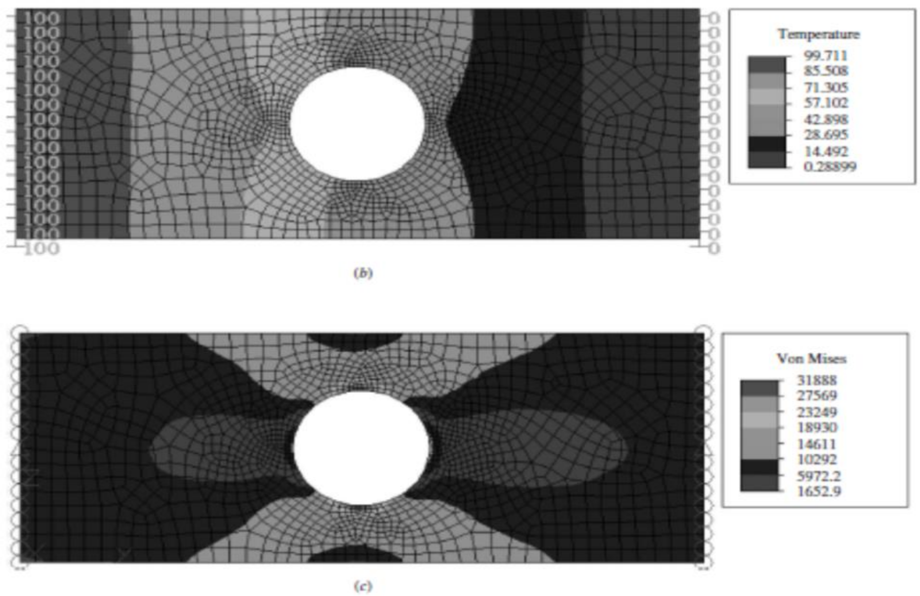


Figure 19–9

(a) Plate supported at ends and maintained at the temperatures shown; (b) steady-state temperature contours; (c) thermal stress contours where the initial temperature of the plate was 0°F.

Figure 19–9*b* shows the temperature contours of the steady-state temperature distribution obtained by the FEA. Using the same elements for a linear stress analysis, where the temperatures were transferred from the heat transfer analysis, Fig. 19–9*c* shows the resulting stress contours. As expected, the maximum compressive stresses occurred at the top and bottom of the hole; with a magnitude of 31.9 kpsi.

❖ Critical Buckling Load

Finite elements can be used to predict the *critical buckling load* for a thin-walled structure. An example was shown in Fig. 4–25 (p. 182). Another example can be seen in Fig. 19–10*a*, which is a thin-walled aluminum beverage can. A specific pressure was applied to the top surface. The bottom of the can was constrained in translation vertically, the center node of the bottom of the can was constrained in translation in all three directions, and one outer node on the can bottom was constrained in translation tangentially. This prevents rigid-body motion, and provides vertical support for the bottom of the can with unconstrained motion of the bottom of the can horizontally. The finite element software returns a value of the load multiplier, which, when multiplied with the total applied force, indicates the critical buckling load.
The buckling mode shape for the buckled beverage can is shown in Fig. 19–10*b*.

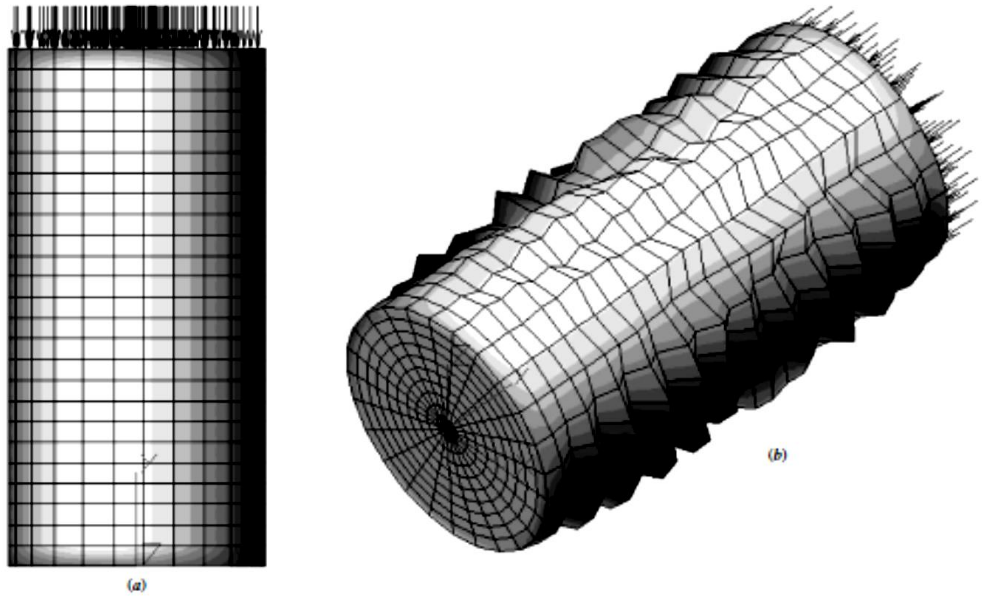


Figure 19–10

(a) Thin-walled aluminum beverage container loaded vertically downward on the top surface; (b) Isometric view of the buckled can (deflections greatly exaggerated).

❖ Vibration Analysis

The design engineer may be concerned as to how a component behaves relative to dynamic input, which results in vibration. For vibration, most finite element packages start with a *modal analysis* of the component. This provides the natural frequencies and mode shapes that the component naturally vibrates at. These are called the eigenvalues and eigenvectors of the component. Next, this solution can be transferred (much the same as for thermal stresses) to solvers for forced vibration analyses, such as frequency response, transient impact, or random vibration, to see how the component's modes behave to dynamic input. The mode shape analysis is primarily based on stiffness and the resulting deflections.

A modal analysis of the beam model without the bearing steps was performed for a 20-element beam model,15 and the 56 384-element brick and tetrahedron model. Needless to say, the beam model took less than 9 seconds to solve, whereas the solid model took *considerably* longer. The first (fundamental) vibration mode was bending and is shown in Fig. 19–11 for both models, together with the respective frequencies.

The difference between the frequencies is about 1.9 percent. Further note that the mode shape is just that, a shape. The actual magnitudes of the deflections are unknown,

only their relative values are known. Thus, any scale factor can be used to exaggerate the view of the deflection shape.

The convergence of the 20-element model was checked by doubling the number of elements. This resulted in no change.

Figure 19–12 provides the frequencies and shapes for the second mode.16 Here, the difference between the models is 3.6 percent.

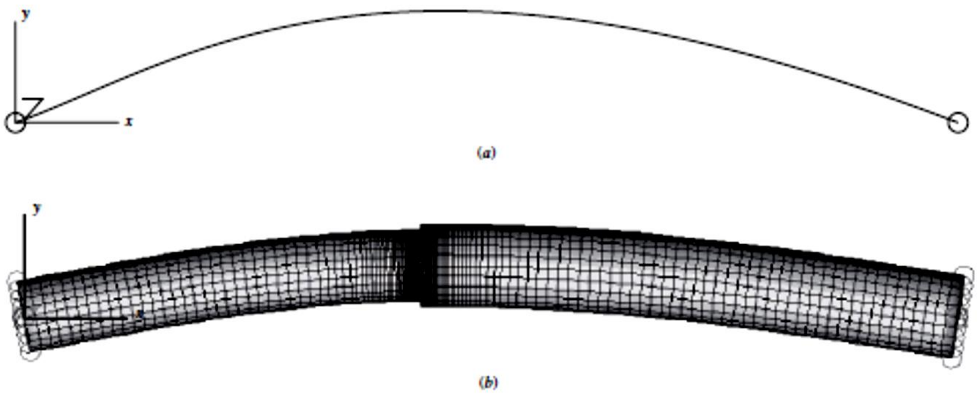


Figure 19–11

First free vibration mode of step beam. (a) Twenty-element beam model, $f_1 = 322$ Hz; (b) 56 384-element brick and tetrahedron model, $f_1 = 316$ Hz.

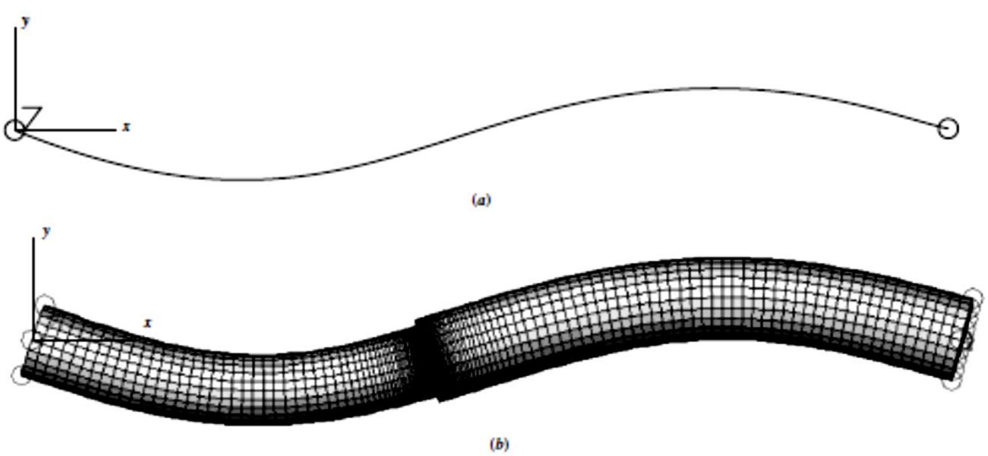


Figure 19–12

Second free-vibration mode of step beam. (a) Twenty-element beam model, $f_2 = 1296$ Hz; (b) 56 384-element brick and tetrahedron model, $f_2 = 1249$ Hz.

As stated earlier, once the mode shapes are obtained, the response of the structure to various dynamic loadings, such as harmonic, transient, or random input, can be obtained. This is accomplished by using the mode shapes together with modal superposition. The method is called *modal analysis*