3.2. Mallado, Cargas y Condiciones de Contorno.

En esta sección se proporcionan algunas breves y sencillas recomendaciones sobre el uso de los programas comerciales de elementos finitos. Seguidamente se indica de forma genérica, dependiendo de la forma geométrica del modelo que se pretende analizar, en qué lugares los elementos deberían ser más pequeños. Se comenta que los elementos que se utilicen tienen que tener una relación de aspecto adecuada, debiéndose evitar el uso de elementos distorsionados. Se indica que si no se controla con cuidado la generación de la malla con estos programas, es muy habitual que aparezcan elementos distorsionados. Para orientar sobre el uso de los distintos tipos de elementos, se indica según la forma geométrica del elemento, cuales son más recomendables, si es posible utilizarlos. Con el fin de insistir sobre el tema de la aplicación de las cargas, se comentan dos procedimientos inmediatos de conversión de cargas distribuidas a cargas nodales equivalentes: el método NODO a NODO; y el ELEMENTO a ELEMENTO. Seguidamente se procede a tratar el tema de las condiciones de contorno. Se distingue entre condiciones de contorno ESENCIALES y NATURALES. Se insiste especialmente en lo que se denomina “supresión de modos de cuerpo rígido”, tanto en problemas bidimensionales como en problemas tridimensionales, proporcionando ejemplos. Con el fin de simplificar los modelos de elementos finitos que se utilicen se comenta la posibilidad de aprovechar las simetrías y antisimétrias que existan, indicando como definir las condiciones de contorno y las cargas en cada caso. Se comentan las que se denominan: simetría reflexiva, simetría rotacional, simetría dihedrica, y simetría translacional, así como la axisimetría.

Por ello facilitamos completo el Tema 8 del Curso Introductorio al Método de los Elementos Finitos que se explica en le Universidad de Colorado en Boulder, bajo la dirección del Prof. Carlos A. Felippa.

CHAPTER 8. Simulación por el MEF: Introducción, Cargas y Condiciones de Contorno.
Carlos A. Felippa.

Al final del capítulo se proponen como ejercicios los siguientes: (1) Dado un modelo geométrico bidimensional, en el que están indicadas las condiciones de carga y los apoyos o condiciones de contorno, se trata de indicar en qué lugares será necesario utilizar una malla de elementos finitos más densa y porque razón; (2) Dada una malla de elementos definida en un modelo bidimensional, en el que existe una zona de transición sin mallar, se trata de proponer posibles mallas para esa zona, de tal forma que sea coherente; (3) dada una distribución lineal de carga, en un modelo de elementos finitos de un problema bidimensional, definido en base a cuadriláteros de cuatro nodos, se trata de calcular las cargas nodales equivalentes mediante los dos procedimientos comentados; (4) dados varios problemas bidimensionales, en los que se proporciona la forma geométrica del mismo, y las cargas aplicadas, se trata de identificar las líneas de simetría y antisimetría que existen, si es posible o no aprovechar su existencia para reducir el tamaño del problema a mallar, y de proponer una posible malla para cada caso, indicando las condiciones de contorno y las cargas que se deberían considerar para resolver cada problema adecuadamente.

8

FEM Modeling: Mesh, Loads and BCs

8–1

TABLE OF CONTENTS

This Chapter continues the exposition of finite element modeling principles. After some general recommendations, it provides guidelines on layout of finite element meshes, conversion of distributed loads to node forces, and how to handle the simplest forms of support boundary conditions. The following Chapters deal with more complicated forms of boundary conditions called multifreedom constraints.

The presentation is "recipe oriented" and illustrated by specific examples. All examples are from structural mechanics; most of them are two-dimensional. No attempt is given at a rigorous justification of rules and recommendations, because that would require mathematical tools beyond the scope of this course.

§8.1. GENERAL RECOMMENDATIONS

The general rules that should guide you in the use of commercial or public FEM packages, are:

- Use the *simplest* type of finite element that will do the job.
- *Never, never, never* mess around with complicated or special elements, unless you are *absolutely sure* of what you are doing.
- Use the *coarsest mesh* you think will capture the dominant physical behavior of the physical system, particularly in *design* applications.

Three word summary: *keep it simple*. Initial FE models may have to be substantially revised to accommodate design changes, and there is little point in using complicated models that will not survive design iterations. The time for refined models is when the design has stabilized and you have a better view picture of the underlying physics, possibly reinforced by experiments or observation.

§8.2. GUIDELINES ON ELEMENT LAYOUT

The following guidelines are stated for structural applications. As noted above, they will be often illustrated for two-dimensional meshes of continuum elements for ease of visualization.

§8.2.1. Mesh Refinement

Use a relatively fine (coarse) discretization in regions where you expect a high (low) *gradient* of strains and/or stresses. Regions to watch out for high gradients are:

- Near entrant corners, or sharply curved edges.
- In the vicinity of concentrated (point) loads, concentrated reactions, cracks and cutouts.
- In the interior of structures with abrupt changes in thickness, material properties or cross sectional areas.

The examples in Figure 8.1 illustrate some of these "danger regions." Away from such regions one can use a fairly coarse discretization within constraints imposed by the need of representing the structural geometry, loading and support conditions reasonably well.

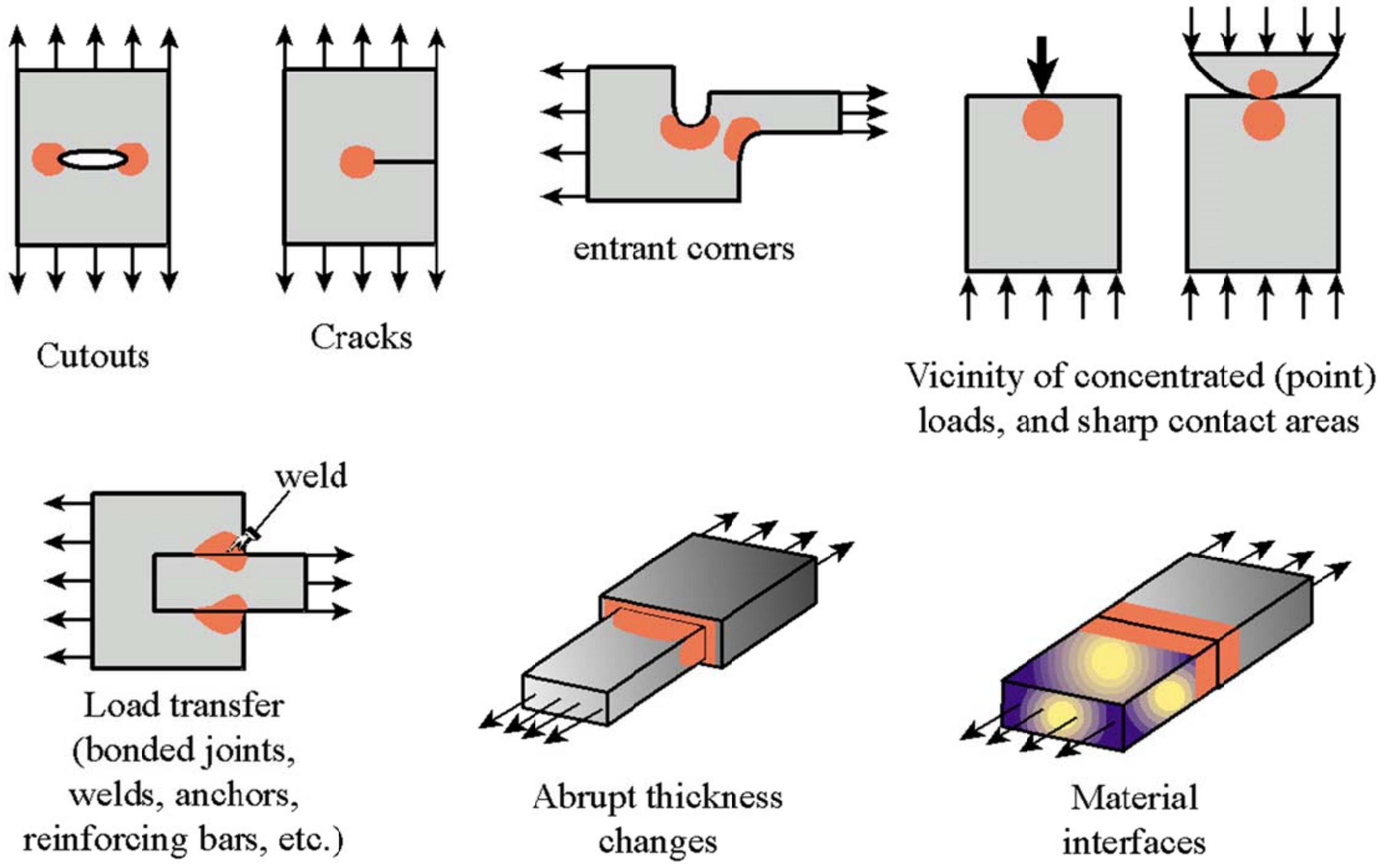

Figure 8.1. Some situations where a locally refined finite element discretization (in the red-colored areas) is recommended.

§8.2.2. Element Aspect Ratios

When discretizing two and three dimensional problems, try to avoid finite elements of high aspect ratios: elongated or "skinny" elements, such as the ones illustrated on the right of Figure 8.2. (The aspect ratio of a two- or three-dimensional element is the ratio between its largest and smallest dimension.)

As a rough guideline, elements with aspect ratios exceeding 3 should be viewed with caution and those exceeding 10 with alarm. Such elements will not necessarily produce bad results — that depends on the loading and boundary conditions of the problem — but do introduce the potential for trouble.

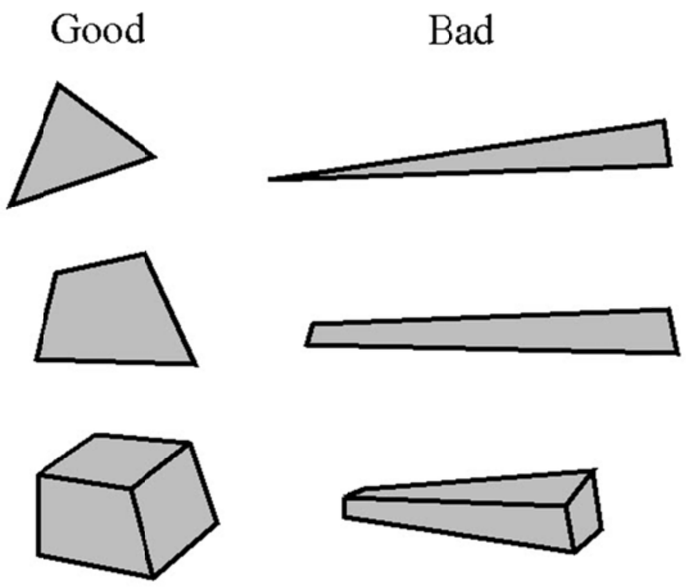

Figure 8.2. Elements with good and bad aspect ratios.

REMARK 8.1

In many "thin" structures modeled as continuous bodies the appearance of "skinny" elements is inevitable on account of computational economy reasons. An example is provided by the three-dimensional modeling of layered composites in aerospace and mechanical engineering problems.

§8.2.3. Physical Interfaces

A physical interface, resulting from example from a change in material, should also be an interelement boundary. That is, *elements must not cross interfaces*. See Figure 8.3.

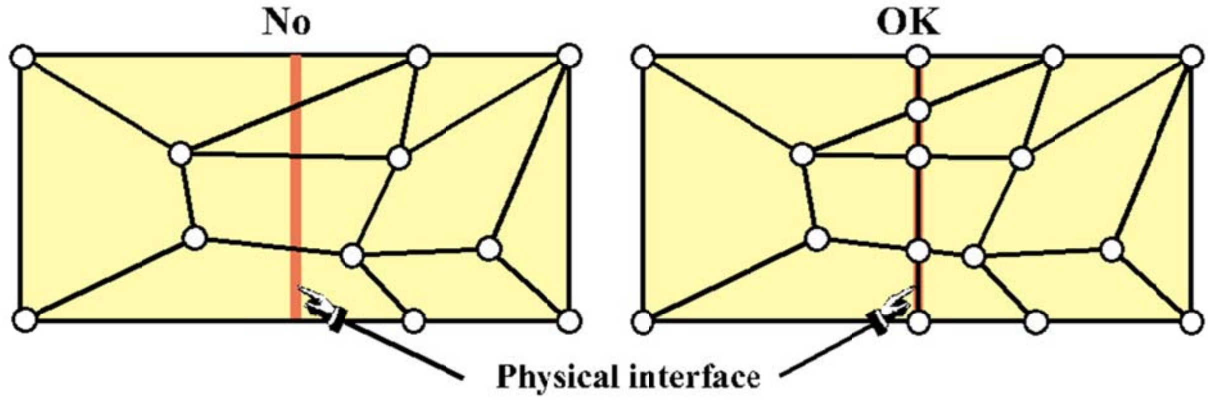

Figure 8.3. llustration of the rule that elements should not cross material interfaces.

§8.2.4. Preferred Shapes

In two-dimensional FE modeling, if you have a choice between triangles and quadrilaterals with similar nodal arrangement, prefer quadrilaterals. Triangles are quite convenient for mesh generation, mesh transitions, rounding up corners, and the like. But sometimes triangles can be avoided altogether with some thought. One of the homework exercises is oriented along these lines.

In three dimensional FE modeling, prefer strongly bricks over wedges, and wedges over tetrahedra. The latter should be used only if there is no viable alternative.[1] The main problem with tetrahedra and wedges is that they can produce wrong stress results even if the displacement solution looks reasonable.

§8.3. DIRECT LUMPING OF DISTRIBUTED LOADS

In practical structural problems, distributed loads are more common than concentrated (point) loads.[2] Distributed loads may be of surface or volume type.

Distributed surface loads (called surface tractions in continuum mechanics) are associated with actions such as wind or water pressure, lift in airplanes, live loads on bridges, and the like. They are measured in force per unit area.

Volume loads (called body forces in continuum mechanics) are associated with own weight (gravity), inertial, centrifugal, thermal, prestress or electromagnetic effects. They are measured in force per unit volume.

A derived type: line loads, result from the integration of surface loads along one transverse direction, or of volume loads along two transverse directions. Line loads are measured in force per unit length.

Whatever their nature or source, distributed loads *must be converted to consistent nodal forces* for FEM analysis. These forces eventually end up in the right-hand side of the master stiffness equations.

The meaning of "consistent" can be made precise through variational arguments, by requiring that the distributed loads and the nodal forces produce the same external work. Since this requires

[1] Unfortunately, many existing space-filling automatic mesh generators in three dimensions produce tetrahedral meshes. There are generators that try to produce bricks, but these often fail in geometrically complicated regions.

[2] In fact, one of the objectives of a good structural design is to avoid or alleviate stress concentrations produced by concentrated forces.

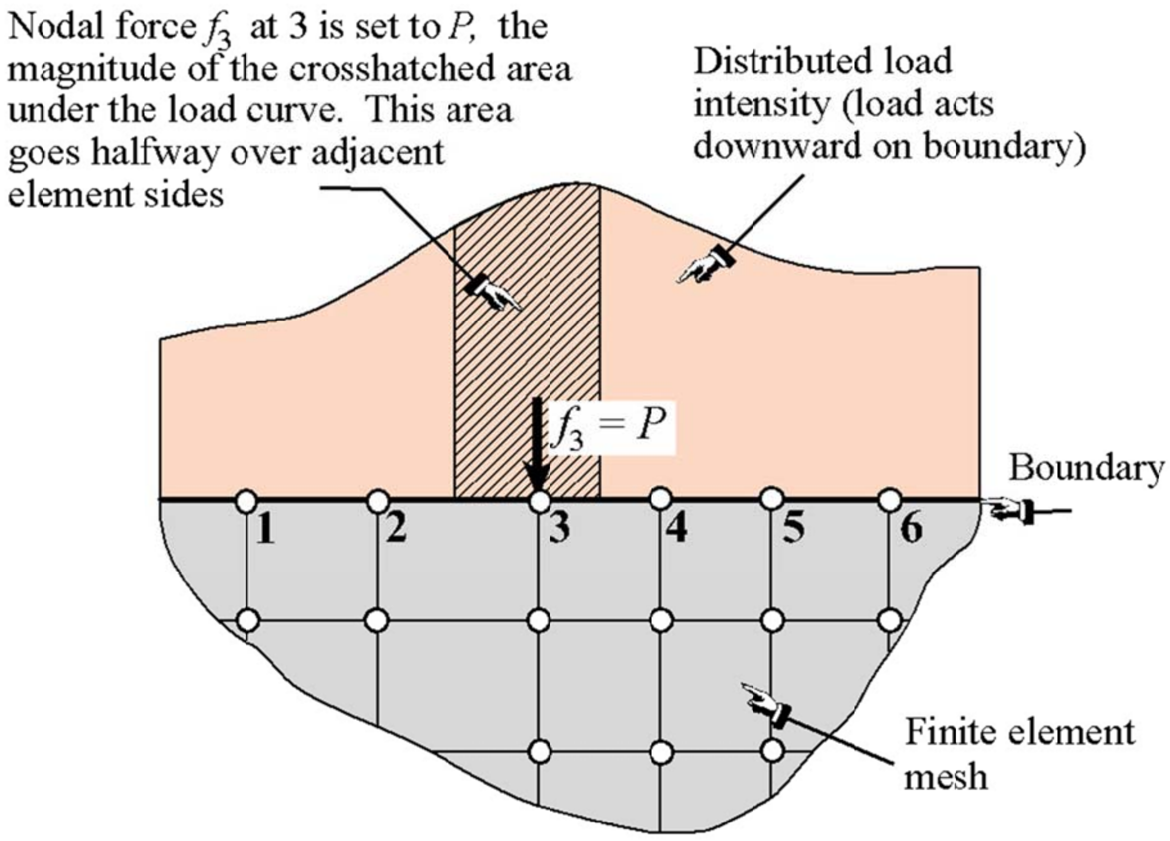

Figure 8.4. NbN direct lumping of distributed load, illustrated for a 2D problem.

the introduction of external work functionals, the topic is deferred to Part II. However, a simpler approach called *direct load lumping*, or simply *load lumping*, is often used by structural engineers in lieu of the more mathematically impeccable but complicated variational approach. Two variants of this technique are described below for distributed surface loads.

§8.3.1. Node by Node (NbN) Lumping

The node by node (NbN) lumping method is graphically explained in Figure 8.4. This example shows a distributed surface loading acting normal to the straight boundary of a two-dimensional FE mesh. (The load is assumed to have been integrated through the thickness normal to the figure, so it is actually a line load measured as force per unit length.)

The procedure is also called *tributary region* or *contributing region* method. For the example of Figure 8.4, each boundary node is assigned a *tributary region* around it that extends halfway to the adjacent nodes. The force contribution P of the cross-hatched area is directly assigned to node 3.

This method has the advantage of not requiring the computation of centroids, as required in the EbE technique discussed in the next subsection. For this reason it is often preferred in hand computations. It can be extended to three-dimensional meshes as well as volume loads.[3] It should be avoided, however, when the applied forces vary rapidly (within element length scales) or act only over portions of the tributary regions.

§8.3.2. Element by Element (EbE) Lumping

In this variant the distributed loads are divided over element domains. The resultant load is assigned to the centroid of the load diagram, and apportioned to the element nodes by statics. A node force

[3] The computation of tributary areas and volumes can be done through the so-called Voronoi diagrams. This is an advanced topic in computational geometry and thus not treatede here.

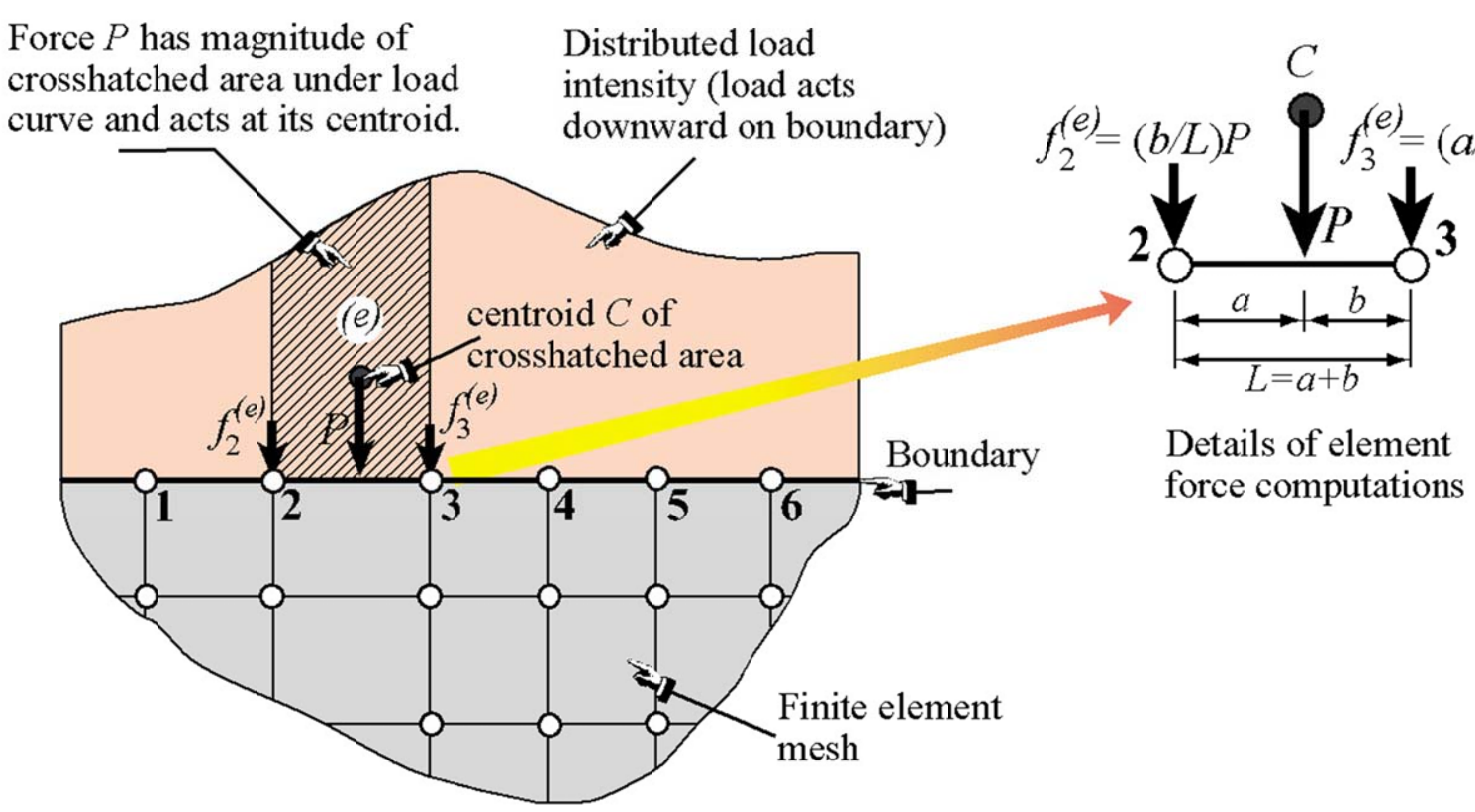

Figure 8.5. EbE direct lumping of distributed load, illustrated for a 2D problem.

is obtained by adding the contributions from all elements meeting at that node. The procedure is illustrated in Figure 8.5, which shows details of the computation over segment 2-3. The total force at node 3, for instance, would be that contributed by segments 2-3 and 3-4.

If applicable, the EbE procedure is more accurate than NbN lumping. In fact it agrees with the consistent node lumping for simple elements that possess only corner nodes. In those cases it is not affected by the sharpness of the load variation and can be even used for point loads that are not applied at the nodes.

The procedure is not applicable if the centroidal resultant load cannot be apportioned by statics. This happens if the element has midside faces or internal nodes in addition to corner nodes, or if it has rotational degrees of freedom. For those elements the variational-based consistent approach covered in Part II is preferable.

§8.4. BOUNDARY CONDITIONS

The key distinction between *essential* and *natural* boundary conditions (BC) was introduced in the previous Chapter. The distinction is explained in Part II from a variational standpoint. In this Chapter we discuss next the simplest *essential* boundary conditions in structural mechanics from a physical standpoint. This makes them relevant to problems with which a structural engineer is familiar. Because of the informal setting, the ensuing discussion relies heavily on examples.

In structural problems formulated by the DSM, the recipe of §7.7.1 that distinguishes between essential and natural BC is: if it directly involves the nodal freedoms, such as displacements or rotations, it is essential. Otherwise it is natural. Conditions involving applied loads are natural. Essential BCs take precedence over natural BCs.

The simplest essential boundary conditions are support and symmetry conditions. These appear in many practical problems. More exotic types, such as multifreedom constraints, require more

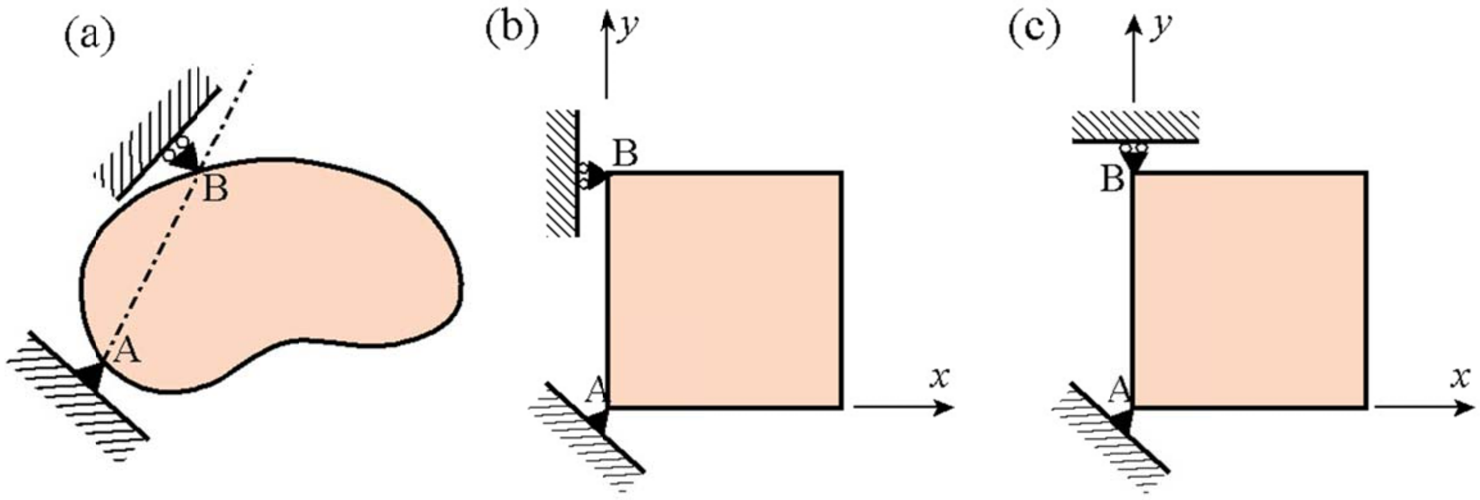

Figure 8.6. Examples of restraining a body against two-dimensional rigid body motions.

advanced mathematical tools and are covered in the next two Chapters.

§8.5. SUPPORT CONDITIONS

Supports are used to restrain structures against relative rigid body motions. This is done by attaching them to Earth ground (through foundations, anchors or similar devices), or to a "ground structure" which is viewed as the external environment.[4] The resulting boundary conditions are often called *motion constraints*. In what follows we analyze two- and three-dimensional motions separately.

§8.5.1. Supporting Two Dimensional Bodies

Figure 8.6 shows two-dimensional bodies that displace in the plane of the paper. If a body is not restrained, an applied load will cause infinite displacements. Regardless of the loading conditions, the structure must be restrained against two translations and one rotation. Consequently the minimum number of constraints that has to be imposed in two dimensions is *three*.

In Figure 8.6, support A provides *translational* restraint, whereas support B, together with A, provides *rotational* restraint. In finite element terminology, we say that we *delete* (fix, remove, preclude) all translational displacements at point A, and that we delete the translational degree of freedom directed along the normal to the AB direction at point B. This body is free to distort in any manner without the supports imposing any displacement constraints.

Engineers call A and B *reaction-to-ground points*. This means that if the supports are conceptually removed, the applied loads are automatically balanced by reactive forces at points A and B, in accordance with Newton's third law. Additional freedoms may be removed to model greater restraint by the environment. However, Figure 8.6(a) does illustrate the *minimal* number of constraints.

Figure 8.6(b) shows a simplified version of Figure 8.6(a). Here the line AB is parallel to the global y axis. We simply delete the x and y translations at point A, and the x translation at point B. If the roller support at B is modified as in 8.6(c), it becomes ineffective in constraining the infinitesimal rotational motion about point A because the rolling direction is normal to AB. The configuration of 8.6(c) is called a *kinematic mechanism*, and will be flagged by a singular modified stiffness matrix.

[4] For example, the engine of a car is attached to the vehicle frame through mounts. The car frame becomes the "ground structure," which moves with respect to Earth ground.

§8.5.2. Supporting Three Dimensional Bodies

Figure 8.7 illustrates the extension of the freedom deletion concept to three dimensions. The minimal number of freedoms that have to be deleted is now *six* and many combinations are possible. In the example of Figure 8.7, all three degrees of freedom at point A have been deleted to prevent rigid body translations. The x displacement component at point B is deleted to prevent rotation about z, the z component is deleted at point C to prevent rotation about y, and the y component is deleted at point D to prevent rotation about x.

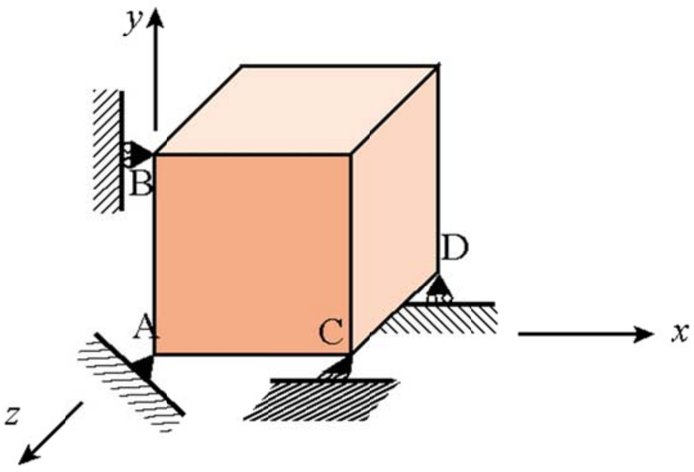

Figure 8.7. Suppressing rigid body motions in a three-dimensional body.

§8.6. SYMMETRY AND ANTISYMMETRY CONDITIONS

Engineers doing finite element analysis should be on the lookout for conditions of *symmetry* or *antisymmetry*. Judicious use of these conditions allows only a portion of the structure to be analyzed, with a consequent saving in data preparation and computer processing time.[5]

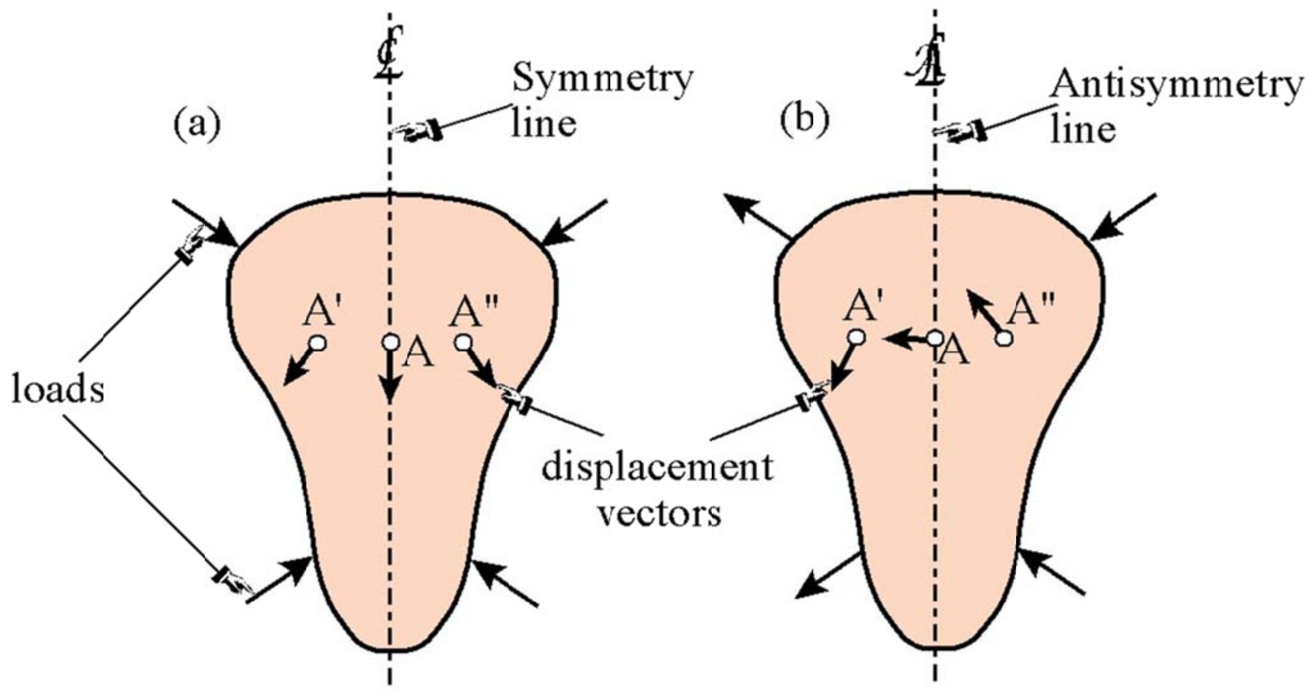

Figure 8.8. Visualizing symmetry and antisymmetry lines.

§8.6.1. Visualization

Recognition of symmetry and antisymmetry conditions can be done by either visualization of the displacement field, or by imagining certain rotational ot reflection motions. Both techniques are illustrated for the two-dimensional case.

A *symmetry line* in two-dimensional motion can be recognized by remembering the "mirror" displacement pattern shown in Figure 8.8(a). Alternatively, a 180° rotation of the body about the symmetry line reproduces exactly the original problem.

[5] Even if the conditions are not explicitly applied through BCs, they provide valuable checks on the computed solution.

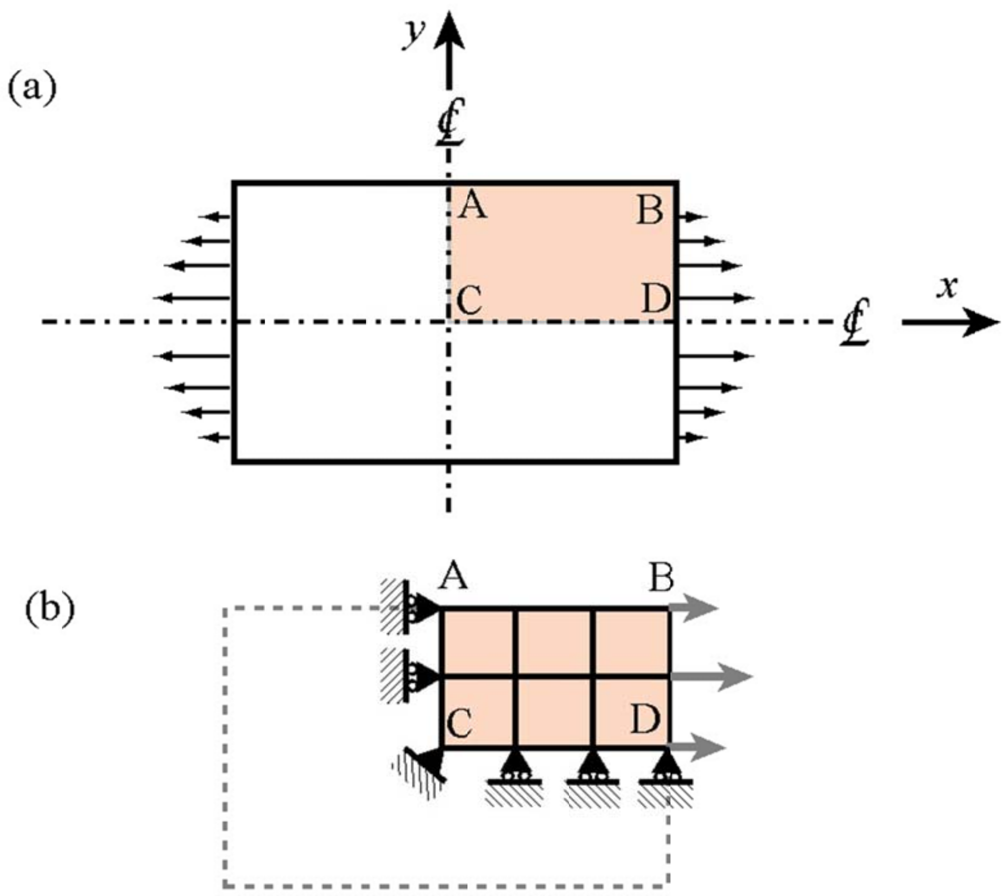

Figure 8.9. A doubly symmetric structure under symmetric loading.

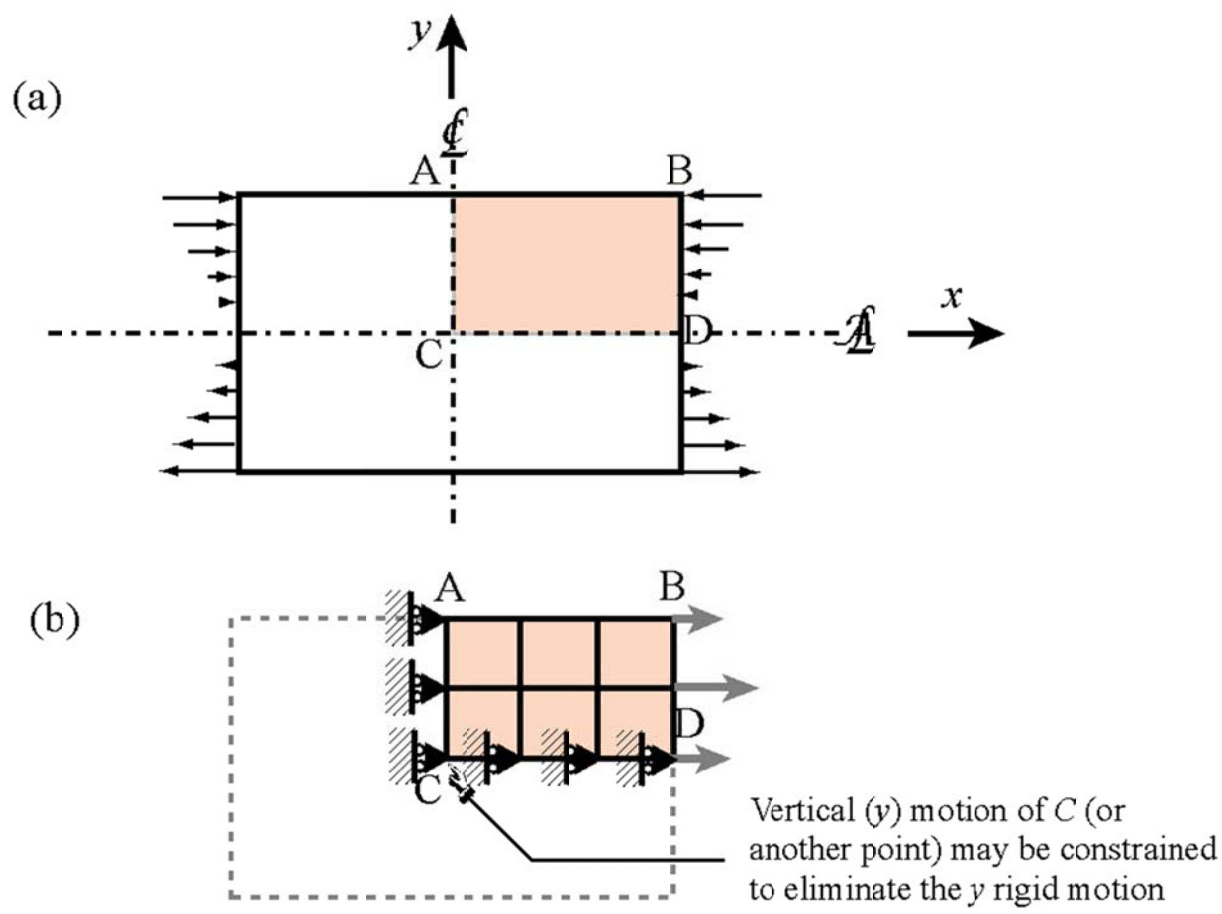

Figure 8.10. A doubly symmetric structure under antisymmetric loading.

An *antisymmetry line* can be recognized by the displacement pattern illustrated in Figure 8.8(b). Alternatively, a 180° rotation of the body about the antisymmetry line reproduces exactly the original problem except that all applied loads are reversed.

Similar recognition patterns can be drawn in three dimensions to help visualization of *planes* of symmetry or antisymmetry. More complex regular patterns associated with *sectorial* symmetry (also called *harmonic* symmetry) and *rotational symmetry* can be treated in a similar manner, but will not be discussed here.

§8.6.2. Effect of Loading Patterns

Although the structure may look symmetric in shape, it must be kept in mind that model reduction can be used only if the loading conditions are also symmetric or antisymmetric.

Consider the plate structure shown in Figure 8.9(a). This structure is symmetrically loaded on the x-y plane. Applying the recognition patterns stated above one concludes that the structure is *doubly symmetric* in both geometry and loading. It is evident that no displacements in the x-direction are possible for any point on the y-axis, and that no y displacements are possible for points on the x axis. A finite element model of this structure may look like that shown in Figure 8.9(b).

On the other hand if the loading is *antisymmetric*, as shown in Figure 8.10(a), then the x axis becomes an *antisymmetry line* because none of the $y = 0$ points can move along the x direction. The boundary conditions to be imposed on the finite element model are also different, as shown in Figure 8.10(b).

REMARK 8.2

For the antisymmetric loading case, one node point has to be constrained against vertical motion. If there are no actual physical supports, the choice is arbitrary and amounts only to an adjustment on the overall (rigid-body) vertical motion. In Figure 8.10(b) the center point C has been chosen to be that vertically-constrained node. But any other node could be selected as well; for example A or D. The important thing is not to overconstrain the structure by applying more than one y constraint.

Notes and Bibliography

FEM modeling rules in most textbooks are "diffuse" if given at all. As noted in Chapter 7, most authors lack practical experience and view FEM as a way to solve BVPs of their own choosing. The rule collection at the start of this Chapter attempts to place key recommendations in one place.

The treatment of BCs tends to be also flaky. A notable exception is Irons and Ahmad [8.1], which is understandable since Irons worked in industry (Rolls-Royce Ltd) before moving to academia.

References

[8.1] Irons, B. M., Ahmad, S., *Techniques of Finite Elements*, Ellis Horwood Ltd, Chichester, UK, 1980.

Homework Exercises for Chapters 7 and 8

FEM Modeling

EXERCISE 8.1

[D:10] The plate structure shown in Figure E8.1 is loaded and deforms in the plane of the figure. The applied load at D and the supports at I and N extend over a fairly narrow area. Give a list of what you think are the likely "trouble spots" that would require a locally finer finite element mesh to capture high stress gradients.

Identify those spots by its letter and a reason. For example, D: vicinity of point load.

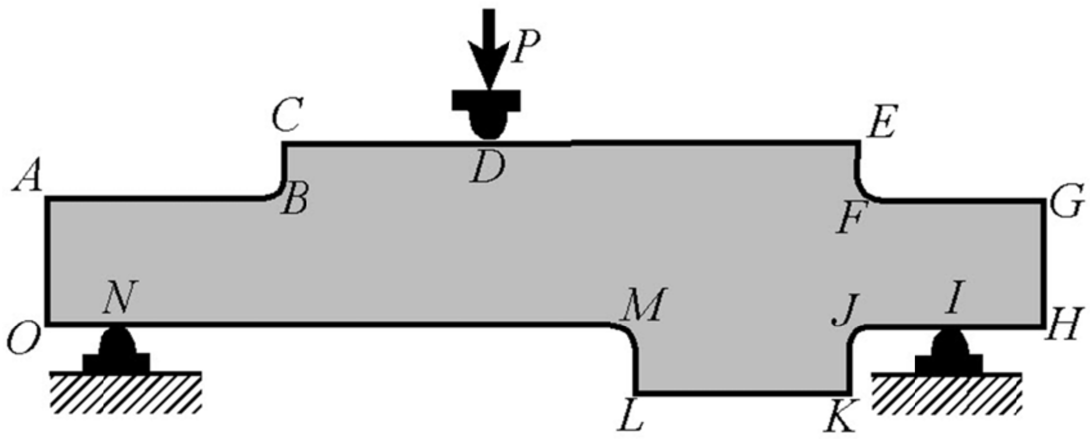

Figure E8.1. Plate structure for Exercise 8.1.

EXERCISE 8.2

[D:15] Part of a two-dimensional FE mesh has been set up as indicated in Figure E8.2. Region $ABCD$ is still unmeshed. Draw a *transition mesh* within that region that correctly merges with the regular grids shown, uses 4-node quadrilateral elements (quadrilaterals with corner nodes only), and *avoids triangles*. *Note*: There are several (equally acceptable) solutions.

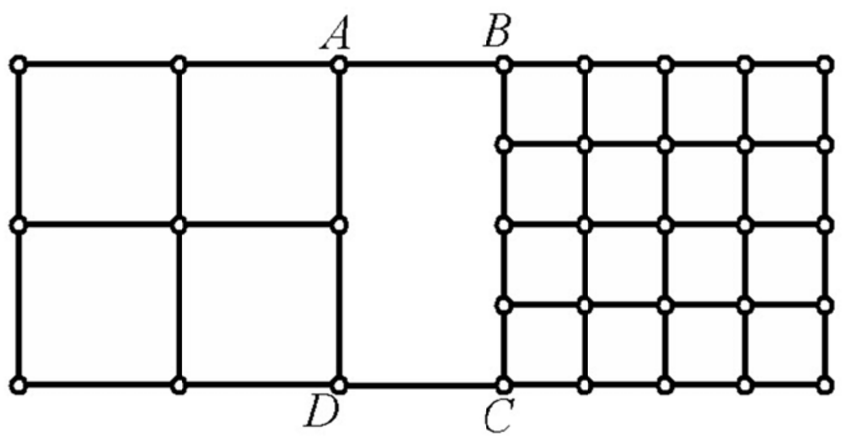

Figure E8.2. Plate structure for Exercise 8.2.

EXERCISE 8.3

[A:15] Compute the "lumped" nodal forces f_1, f_2, f_3 and f_4 equivalent to the linearly-varying distributed surface load q for the finite element layout defined in Figure E8.3. Use both NbN and EbE lumping. For example, $f_1 = 3q/8$ for NbN. Check that $f_1 + f_2 + f_3 + f_4 = 6q$ for both schemes (why?). Note that q is given as a force per unit of vertical length.

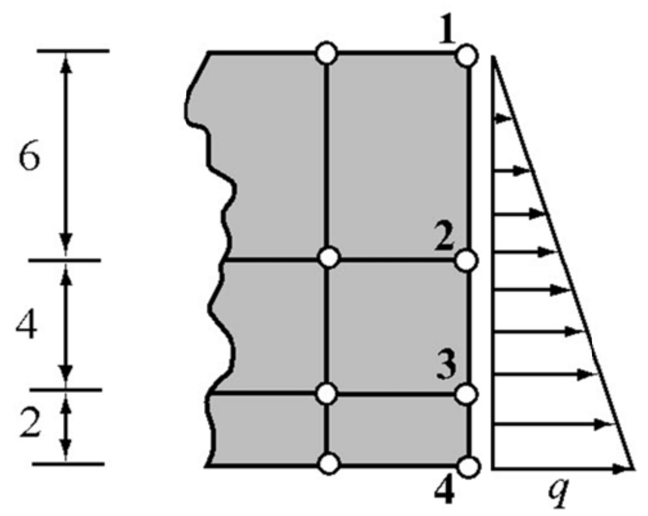

Figure E8.3. Mesh layout for Exercise 8.3.

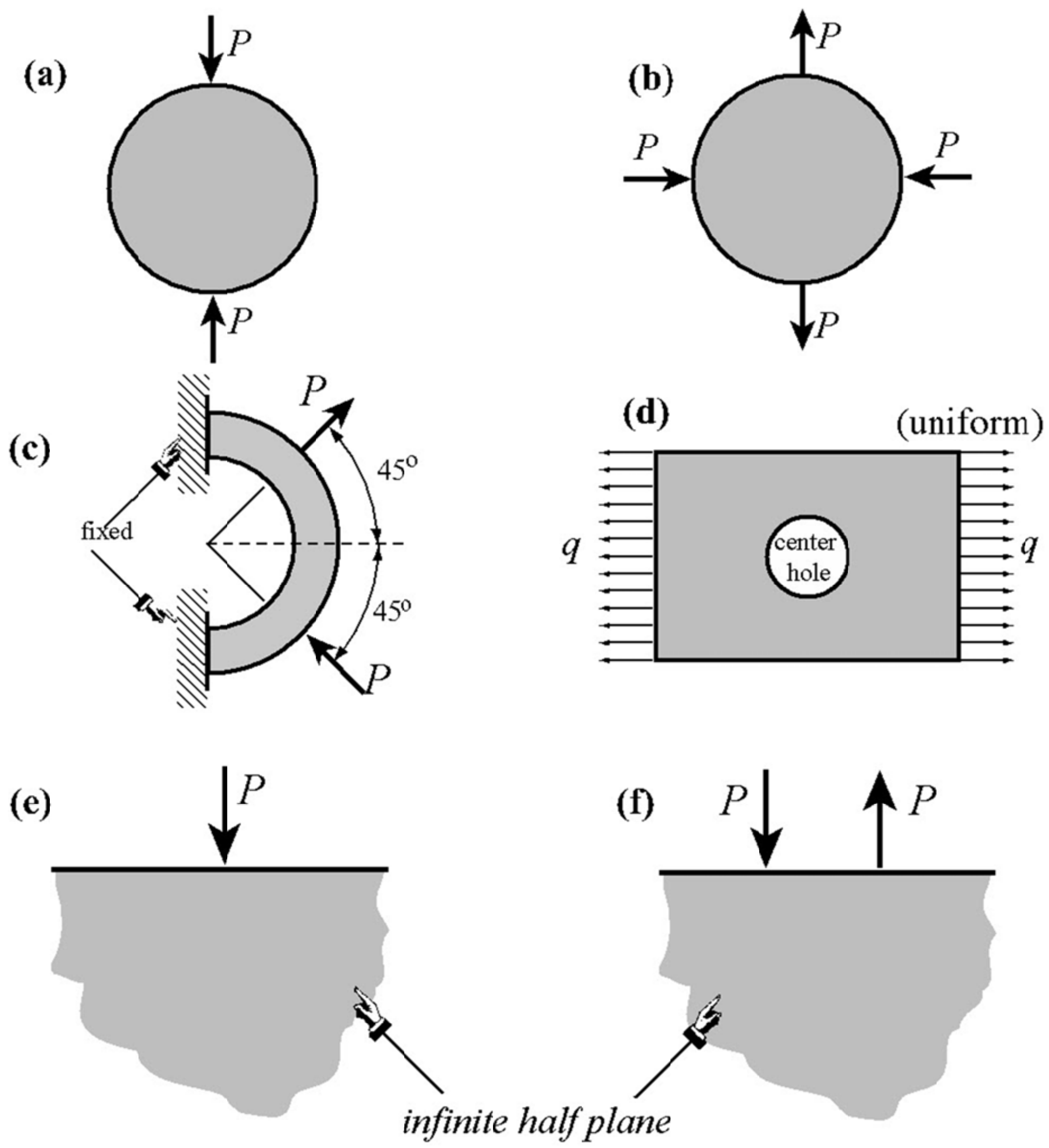

Figure E8.4. Problems for Exercise 8.4.

EXERCISE 8.4

[D:15] Identify the symmetry and antisymmetry lines in the two-dimensional problems illustrated in Figure E8.4. They are: (a) a circular disk under two diametrically opposite point forces (the famous "Brazilian test" for concrete); (b) the same disk under two diametrically opposite force pairs; (c) a clamped semiannulus under

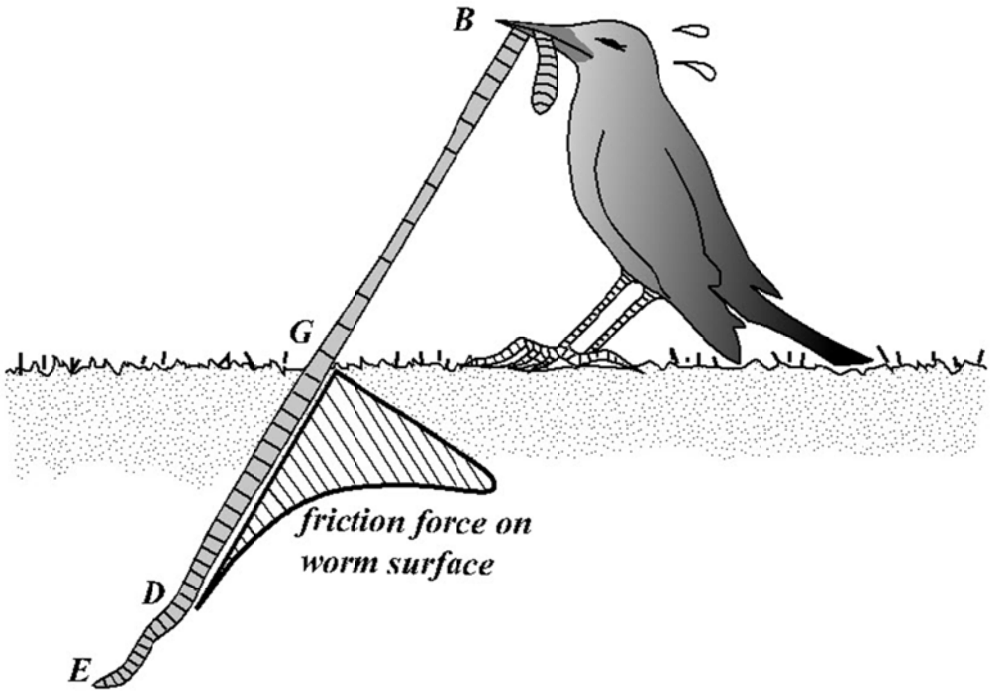

Figure E8.5. The hungry bird.

a force pair oriented as shown; (d) a stretched rectangular plate with a central circular hole. Finally (e) and (f) are half-planes under concentrated loads.[6]

Having identified those symmetry/antisymmetry lines, state whether it is possible to cut the complete structure to one half or one quarter before laying out a finite element mesh. Then draw a coarse FE mesh indicating, with rollers or fixed supports, which kind of displacement BCs you would specify on the symmetry or antisymmetry lines. *Note: Do all sketches on your paper, not on the printed figures.*

EXERCISE 8.5

[D:20] You (a finite element guru) pass away and come back to the next life as an intelligent but hungry bird. Looking around, you notice a succulent big worm taking a peek at the weather. You grab one end and pull for dinner; see Figure E8.5.

After a long struggle, however, the worm wins. While hungrily looking for a smaller one your thoughts wonder to FEM and how the bird extraction process might be modeled so you can pull it out more efficiently. Then you wake up to face this homework question. Try your hand at the following "worm modeling" points.

(a) The worm is simply modeled as a string of one-dimensional (bar) elements. The "worm axial force" is of course constant from the beak B to ground level G, then decreases rapidly because of soil friction (which varies roughly as plotted in the figure above) and drops to nearly zero over DE. Sketch how a good "worm-element mesh" should look like to capture the axial force well.

(b) On the above model, how would you represent boundary conditions, applied forces and friction forces?

(c) Next you want a more refined anaysis of the worm that distinguishes skin and insides. What type of finite element model would be appropriate?

(d) (Advanced) Finally, point out what need to be added to the model of (c) to include the soil as an elastic medium.

Briefly explain your decisions. Dont write equations.

EXERCISE 8.6

[A/D:20] Explain from kinematics why two antisymmetry lines in 2D cannot cross at a finite point. As a

[6] Note that (e) is the famous Flamant's problem, which is important in the 2D design of foundations of civil structures. The analytical solution of (e) and (f) may be found, for instance, in Timoshenko-Goodier's *Theory of Elasticity*, 2nd Edition, page 85ff.

corollary, investigate whether it is possible to have more than one antisymmetry line in a 2D elasticity problem.

EXERCISE 8.7

[A/D:15] Explain from kinematics why a symmetry line and an antisymmetry line must cross at right angles.

EXERCISE 8.8

[A/D:15] A 2D body has $n > 1$ symmetry lines passing through a point C and spanning an angle π/n from each other. This is called *sectorial symmetry* if $n \geq 3$. Draw a picture for $n = 5$, say for a car wheel. Explain why C is fixed.

EXERCISE 8.9

[A/D:25, 5 each] A body is in 3D space. The analogs of symmetry and antisymmetry lines are symmetry and antisymmetry planes, respectively. The former are also called mirror planes.

(a) State the kinematic properties of symmetry and antisymmetric planes, and how they can be identified.

(b) Two symmetry planes intersect. State the kinematic properties of the intersection line.

(c) A symmetry plane and an antisymmetry plane planes intersect. State the kinematic properties of the intersection line. Can the angle between the planes be arbitrary?

(d) Can two antisymmetry planes intersect?

(e) Three symmetry planes intersect. State the kinematic properties of the intersection point.

EXERCISE 8.10

[A:25] A 2D problem is called *periodic* in the x direction if all fields, in particular displacements, repeat upon moving over a distance $a > 0$: $u_x(x + a, y) = u_x(x, y)$ and $u_y(x + a, y) = u_y(x, y)$. Can this situation be treated by symmetry and/or antisymmetry lines?

EXERCISE 8.11

[A:25] Extend the previous exercise to *antiperiodicity*, in which $u_x(x + a, y) = u_x(x, y)$ and $u_y(x + a, y) = -u_y(x, y)$.

EXERCISE 8.12

[A:40] If the world were spatially n-dimensional (meaning it has elliptic metric), how many independent rigid body modes would a body have? (Prove by induction)