7

FEM Modeling: Mesh, Loads and BCs

TABLE OF CONTENTS

§7.1. Introduction

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

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

§7.2. General Recommendations

The general rules that should guide you in the use of commercial or public FEM packages, are[1]

- 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. There is little point in using complicated models that will not survive design iterations. The time for refinement is when the design has stabilized and you have a better picture of the underlying physics, possibly reinforced by experiments or observation.

§7.3. 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.

§7.3.1. Mesh Refinement

Use a relatively fine (coarse) discretization in regions where you expect a high (low) *gradient* of strains and/or stresses.[2] Regions to watch out for high gradients are:

- Near entrant corners (see Remark below) 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 7.1 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.

[1] Paraphrasing the Bellman in The Hunting of the Snark: "what I say three times is true."

[2] *Gradient* is the key word. High gradient means rapid variation. A high value by itself means nothing in this context.

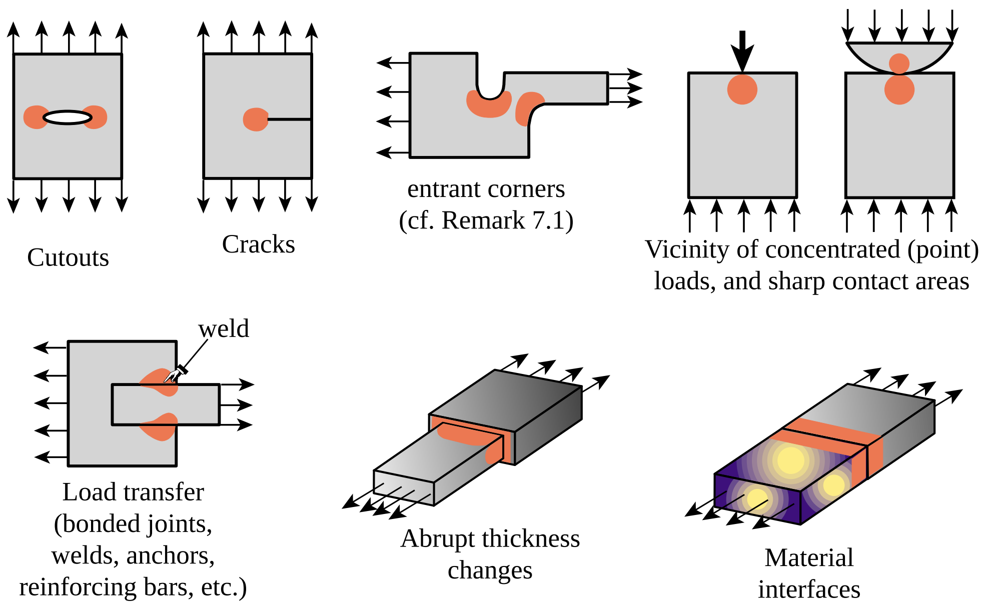


FIGURE 7.1. Some situations where a locally refined finite element discretization (in the red-shaded areas) is recommended.

Remark 7.1. The first bullet above mentions "entrant corner." That is a region where isostatics (principal stress trajectories) "bunch up." For a two-dimensional problem mathematically posed on a singly-connected interior domain, they can be recognized as follows. Traverse the boundary CCW so the body or structure is on your left. When hitting a sharp or rounded corner, look at the angle (positive CCW) formed by the *exterior* normals (the normal going toward your right) before and after, measured *from* before *to* after, and always taking the *positive* value.

If the angle exceeds 180°, it is an entrant corner. [If it is close to 360°, it is a crack tip.] For exterior problems or multiple-connected domains, the definition must be appropriately adjusted.

§7.3.2. 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 7.2. (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.

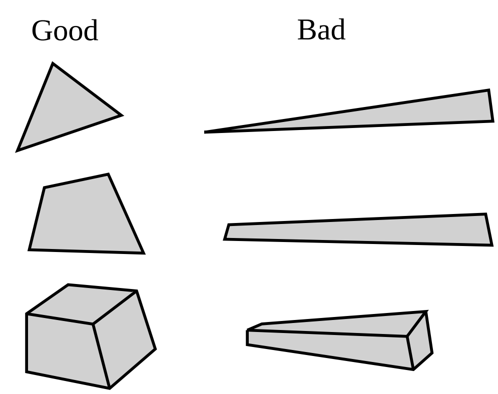


FIGURE 7.2. Elements with good and bad aspect ratios.

Remark 7.2. 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.

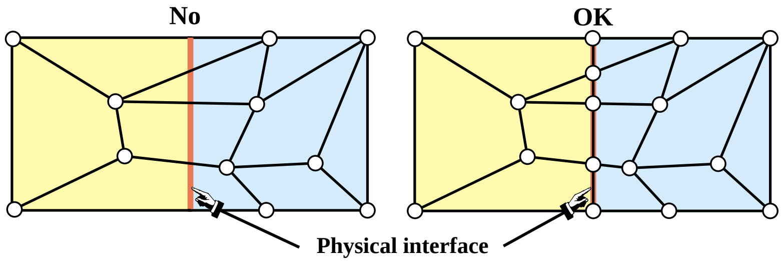


FIGURE 7.3. llustration of the rule that elements should not cross material interfaces.

§7.3.3. 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 7.3.

§7.3.4. Preferred Shapes

In 2D 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 3D FE modeling, prefer strongly bricks over wedges, and wedges over tetrahedra. The latter should be used only if there is no viable alternative.[3] The main problem with tetrahedra and wedges is that they can produce wrong stress results even if the displacement solution looks reasonable.

§7.4. Direct Lumping of Distributed Loads

In practical structural problems, distributed loads are more common than concentrated (point) loads.[4] 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, snow weight on roofs, 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, such as a beam or plate thickness, or of volume loads along two transverse directions, such as a bar or beam area. 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 end up in the right-hand side of the master stiffness equations.

---

[3] 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.

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

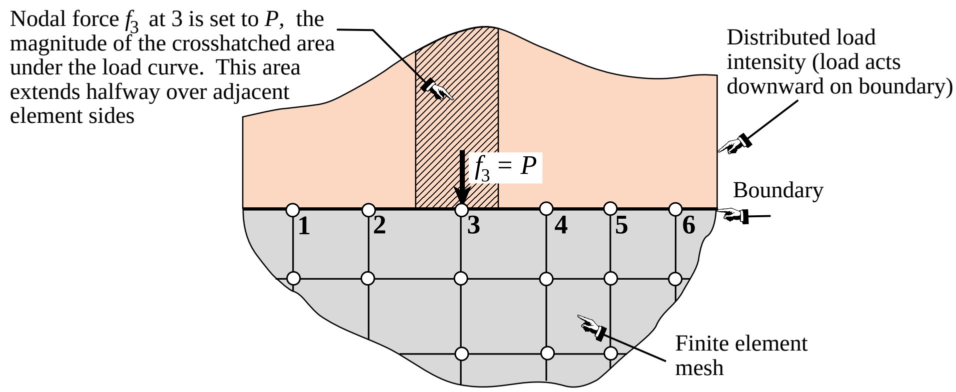


FIGURE 7.4. NbN direct lumping of distributed line load, illustrated for a 2D problem.

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 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 mathematically impeccable but complicated variational approach. Two variants of this technique are described below for distributed surface loads.

§7.4.1. Node by Node Lumping

The node by node (NbN) lumping method is graphically explained in Figure 7.4. 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 7.4, each boundary node is assigned a *tributary region* around it that extends halfway to 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 error-prone computation of centroids, as needed in the EbE technique discussed below. For this reason it is often preferred in hand computations.[5] It can be extended to three-dimensional meshes as well as volume loads.[6] It should be avoided, however, when the applied forces vary rapidly (within element length scales) or act only over portions of the tributary regions.

§7.4.2. Element by Element 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 is obtained by adding the contributions from all elements meeting at that node. The procedure is illustrated in Figure 7.5, which shows details of the computation over 2–3. The total force at node 3, for instance, would be that contributed by segments 2–3 and 3–4. For the frequent case in which

---

[5] It has been extensively used in the aircraft industry for smooth-varying pressure loads computations.

[6] The computation of tributary areas and volumes for general 2D and 3D regions can be done through the so-called Voronoi diagrams. This is an advanced topic in computational geometry (see, e.g., [208]) and thus not treated here.

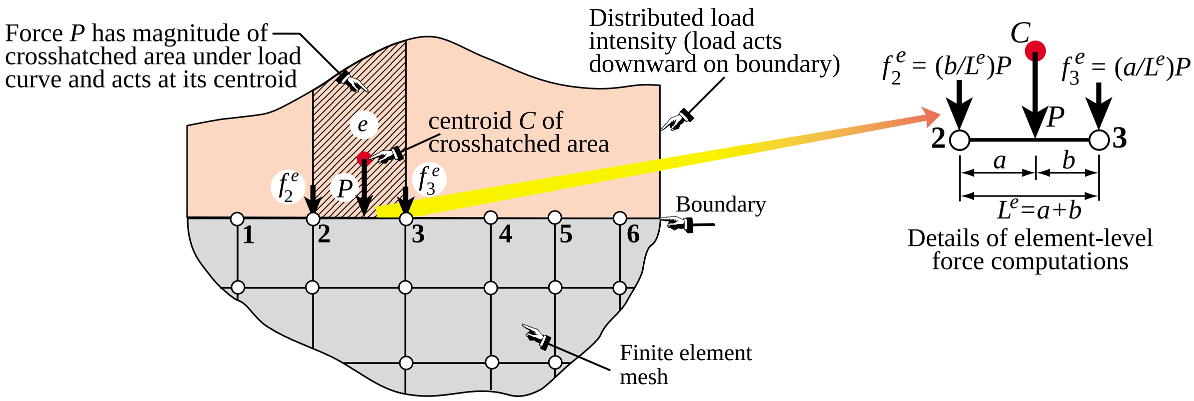


FIGURE 7.5. EbE direct lumping of distributed line load, illustrated for a 2D problem.

the variation of the load over the element is linear (so the area under the load is a trapezoid) the node forces can be computed directly by the formulas given in Figure 7.6.

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

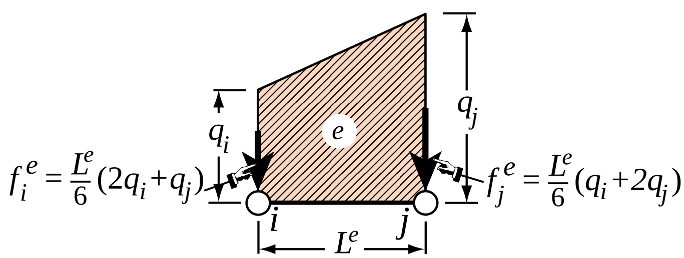


FIGURE 7.6. EbE lumping of linearly varying line load over element.

The EbE 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 and briefly outlined in §7.3.3, is preferable.

Example 7.1. Figures 7.7(a,b) show web-downloaded pictures of the Norfork Dam, a 220-ft high, concrete-built gravity dam. Its typical cross section is shown in 7.7(c). The section is discretized by triangular elements as illustrated in Figure 7.7(d). The dam has a length of 2624 ft and was constructed over the White River in Arkansas over 1941–44.[7] The structure is assumed to be in plane strain. Accordingly the FEM model shown in 7.7(d) is a typical 1-ft slice of the dam and near-field soil.

In the analysis of dam and marine structures, a "wet node" of a FEM discretization is one in contact with the water. The effect of hydrostatic pressure is applied to the structure through nodal forces on wet nodes. The wet nodes for a water head of 180 ft over the riverbed are shown in Figure 7.8(b). Nodes are numbered 1 through 9 for convenience. The pressure in psf (lbs per sq-ft) is $p = 62.4\,d$ where d is the depth in ft. Compute the horizontal hydrostatic nodal forces f_{x1} and f_{x2} using NbN and EbE, assuming that the wet face AB is vertical for simplicity.

Node by Node. For node 1 go halfway to 2, a distance of $(180 - 136)/2 = 22$ ft. The tributary load area is a triangle extending 22 ft along y with bottom pressure $62.4 \times 22 = 1372.8$ psf and unit width normal to

[7] As discussed in **Notes and Bibliography**, this example has historical significance, as the first realistic Civil Engineering structure modeled ca. 1960 by the Finite Element Method, which until then had been largely confined to aerospace.

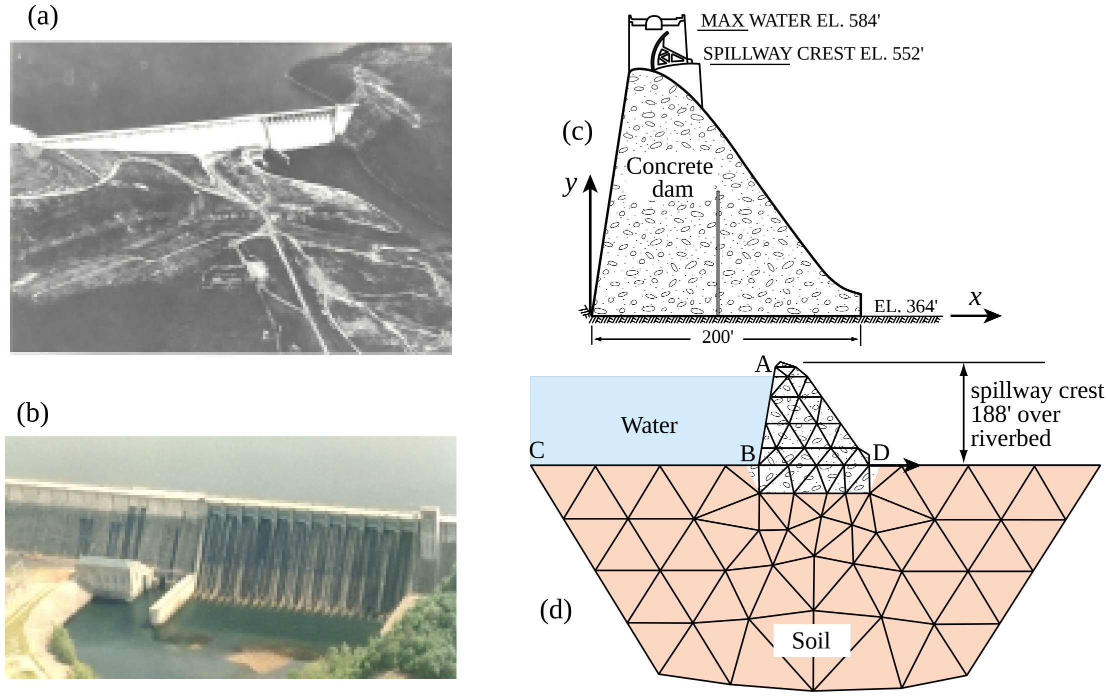


FIGURE 7.7. Norfork Dam: (a,b) pictures; (c) cross section of dam above foundation (line inside dam is a thermally induced crack considered in the 1960 study); (d) coarse mesh including foundation and soil but not crack [146].

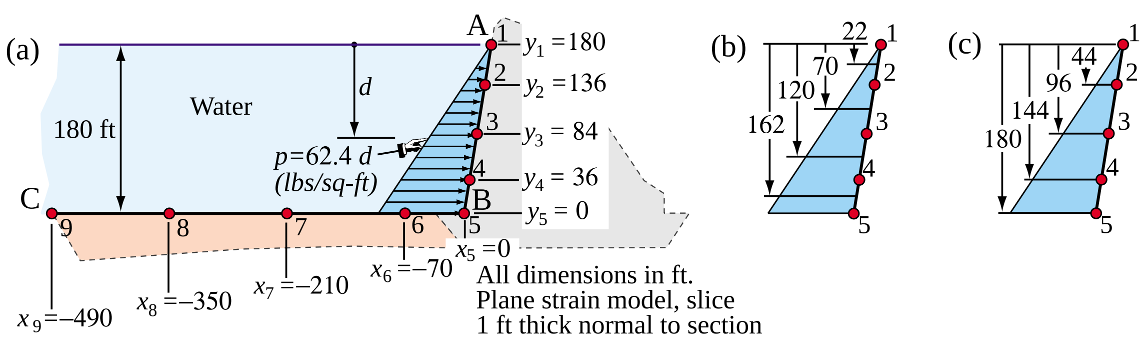


FIGURE 7.8. Norfork Dam example: (a) computation of wet node forces due to hydrostatic pressure; (b) NbN tributary regions; (c) EbE regions.

paper, giving $f_{x1} = \frac{1}{2}22 \times 1372.8 = 15101$ lbs. For node 2 go up 22 ft and down $(136-84)/2 = 26$ ft. The tributary load area is a trapezoid extending $22+26 = 48$ ft vertically, with pressures 1372.8 psf at the top and $62.4 \times 70 = 4368.0$ psf at the bottom. This gives $f_{x2} = 48 \times (1372.8+4368.0)/2 = 137779$ lbs.

Element by Element. It is convenient to pre-compute hydro pressures at wet node levels: $p_1 = 0$, $p_2 = 62.4 \times 44 = 2745.6$ psf, $p_3 = 62.4 \times 96 = 5990.4$. Because the variation of p is linear, the formulas of Figure 7.6 can be applied directly: $f_{x1}^{(1)} = (44/6)(2 \times 0 + 2745.6) = 20134$ lbs, $f_{x2}^{(1)} = (44/6)(0 + 2 \times 2745.6) = 40269$ lbs and $f_{x2}^{(2)} = (52/6)(2 \times 2745.6 + 5990.4) = 99507$ lbs. Adding contributions to nodes 1 and 2: $f_{x1} = f_{x1}^{(1)} = 20134$ lbs and $f_{x2} = f_{x2}^{(1)} + f_{x2}^{(2)} = 40269 + 99507 = 139776$ lbs.

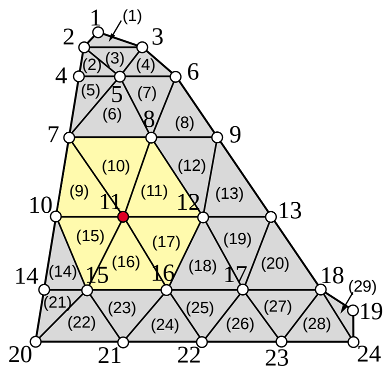
The computations for wet nodes 3 through 9 are left as an exercise.

Example 7.2. Figure 7.9 shows the mesh for the $y > 0$ portion of the gravity dam of the previous example.

(The node numbering is different). The specific weight for concrete of $\gamma = 200$ pcf (pounds per cubic foot). Compute the node force f_{y11} due to own weight.

For this calculation NbN is unwieldy because computation of the nodal tributary region requires construction of a Voronoi diagram. (Furthermore for constant specific weight it gives the same answer as EbE.) To apply EbE, select the elements that contribute to 11: the six triangles (9), (10), (11), (15), (16) and (17).

The area of a triangle with corners at $\{x_1, y_1\}$, $\{x_2, y_2\}$ and $\{x_3, y_3\}\}$ is given by $A = (x_2 y_3 - x_3 y_2) + (x_3 y_1 - x_1 y_3) + (x_1 y_2 - x_2 y_1)$. Applying this to the geometry of the figure one finds that the areas are $A^{(9)} =$, $A^{(10)} =$, $A^{(11)} =$, $A^{(15)} =$, $A^{(16)} =$, and $A^{(17)} =$. The weight forces on each element are $W^{(9)} = \gamma h$, $W^{(10)} = \gamma h$, $W^{(11)} = \gamma h$, $A^{(15)} = \gamma h$, $A^{(16)} = \gamma h$, and $A^{(17)} = \gamma h$, where h is the thickness normal to the figure (1 ft here).



Node	x	y
7	*.00	152.00
8	*.00	152.00
9	*.00	152.00
10	*.00	94.00
11	*.00	94.00
12	*.00	94.00
13	*.00	94.00
14	*.00	40.00
15	*.00	40.00
16	*.00	40.00
17	*.00	40.00
18	*.00	40.00

FIGURE 7.9. Computation of weight force at node 11 of gravity dam example of Figure 7.7(d).

For uniform element thickness and specific weight, one third of each force goes to each element corner. Thus node 11 receives

$$f_{y11} = \tfrac{1}{3}(W^{(9)} + W^{(10)} + W^{(11)} + W^{(17)} + W^{(18)}) = \tag{7.1}$$

(example incomplete, TBF)

§7.4.3. *Weighted Lumping

The NbN and EbE methods are restricted to simple elements, specifically those with corner nodes only. We outline here a general method that works for more complicated models. The mathematical justification requires energy theorems covered in Part II. Thus at this stage the technique is merely presented as a recipe.

To fix the ideas consider again the 2D situation depicted in Figures 7.4 and 7.5. Denote the distributed load by $q(x)$. We want to find the lumped force f_n at an interior node n of coordinate x_n. Let the adjacent nodes be $n-1$ and $n+1$, with coordinates x_{n-1} and x_{n+1}, respectively. Introduce a *weight function* $W_n(x)$ with properties to be specified below. The lumped force is given by

$$f_n = \int_{x_{n-1}}^{x_{n+1}} W_n(x)\, q(x)\, dx \tag{7.2}$$

For this formula to make sense, the weight function must satisfy several properties:

1. Unit value at node n: $W_n(x_n) = 1$.
2. Vanishes over any element not pertaining to n: $W_n = 0$ if $x \le x_{n-1}$ or $x \ge x_{n+1}$
3. Gives the same results as NbN or EbE for constant q over elements with corner nodes only.

Both NbN and EbE are special cases of (7.2). For NbN pick

$$W_n(x) = 1 \quad \text{if} \quad \tfrac{1}{2}(x_{n-1} + x_n) \le x \le \tfrac{1}{2}(x_n + x_{n+1}), \qquad w_n(x) = 0 \;\; \text{otherwise.} \tag{7.3}$$

For EbE pick

$$W_n(x) = \begin{cases} 1 - \dfrac{x - x_{n-1}}{x_n - x_{n-1}} & \text{if } \; x_{n-1} \le x \le x_n, \\ 1 - \dfrac{x - x_n}{x_{n+1} - x_n} & \text{if } \; x_n \le x \le x_{n+1}, \\ 0 & \text{otherwise.} \end{cases} \tag{7.4}$$

These particular weight functions are depicted in Figure 7.10.

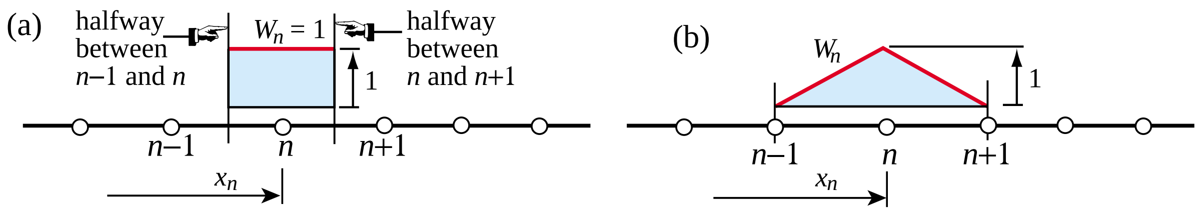


FIGURE 7.10. Weight functions corresponding to: (a) NbN lumping, and (b) EbE lumping.

§7.4.4. *Energy Consistent Lumping

The rule (7.2) can be justified from the standpoint of the Principle of Virtual Work. Let δu_n be a virtual node displacement paired to f_n. Take $\delta u_n W_n(x)$ to be the associated displacement variation. The external virtual work of $q(x)$ is $\int q(x) W_n(x)\, \delta u_n\, dx$ extended over the portion where $W_n \neq 0$. Equating this to $f_n\, \delta u_n$ and cancelling δu_n from both sides yields (7.2).

If W_n is the *trial displacement function* actually used for the development of element equations the lumping is called *energy consistent*, or *consistent* for short.

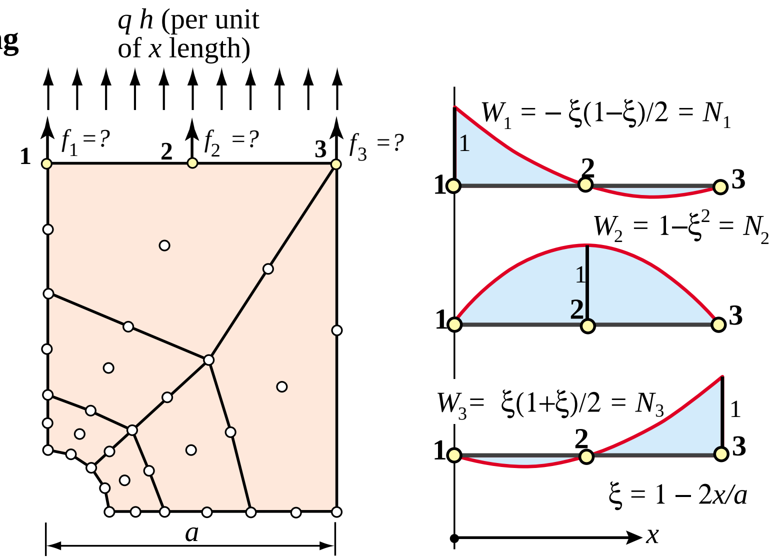


FIGURE 7.11. Example of consistent load lumping.

(As will be seen later, trial functions are the union of shape functions over the patch of all elements connected to node n.) This important technique is studied in Part II of the course after energy methods are introduced.

Example 7.3. Conside the mesh of 9-node quadrilaterals shown in Figure 7.11. (This is later used as a benchmark problem in Chapter 27.) The upper plate edge 1–3 is subject to a uniform normal load qh per unit length, where h is the plate thickness. The problem is to compute the node forces f_1, f_2 and f_3. If 1–2 and 2–3 were on two different elements, both NbN and EbE would give $f_1 = f_3 = \frac{1}{4}qha$ and $f_2 = \frac{1}{2}qha$. But this lumping is wrong for an element with midside nodes. Instead, pick the weight functions W_i ($i = 1, 2, 3$) shown on the right of Figure 7.11.

Since there is only one element on the loaded edge, the W_i are actually the quadratic shape functions N_i for a 3-node line element, developed in later Chapters. The dimensionless variable ξ is called an *isoparametric natural coordinate*. Applying the rule (7.2) we get

$$f_1 = \int_0^a W_1(x)\, q\, h\, dx = \int_{-1}^{1} W_1(\xi)\, q\, h\, \frac{dx}{d\xi}\, d\xi = \int_{-1}^{1} -\tfrac{1}{2}\xi(1-\xi) q\, h\, (\tfrac{1}{2}a)\, d\xi = \tfrac{1}{6} qha. \tag{7.5}$$

Similarly $f_2 = \frac{2}{3}qha$ and $f_3 = f_1$. As a check, $f_1 + f_2 + f_3 = qha$, which is the total load acting on the plate edge.

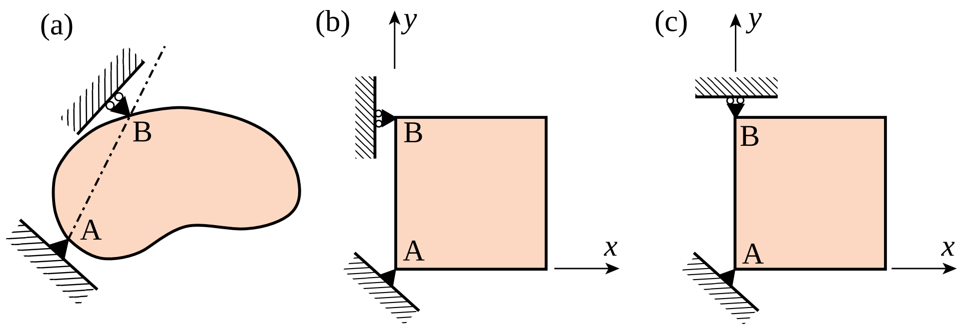


FIGURE 7.12. Suppressing two-dimensional rigid body motions.

§7.5. 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 section we discuss 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 §6.7.1 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 advanced mathematical tools and are covered in the next two Chapters.

§7.6. 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.[8] The resulting boundary conditions are often called *motion constraints*. In what follows we analyze two- and three-dimensional motions separately.

§7.6.1. Supporting Two Dimensional Bodies

Figure 7.12 shows two-dimensional bodies that move in the plane of the paper. If a body is not restrained, an applied load will cause infinite displacements. Regardless of loading conditions, the body must be restrained against two translations along x and y, and one rotation about z. Thus the minimum number of constraints that has to be imposed in two dimensions is *three*.

In Figure 7.12, 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

[8] 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, as Earth rotates and moves through space, etc.

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 deformation constraints.

Engineers call A and B *reaction-to-ground points*. This means that if the supports are conceptually removed, applied loads are automatically balanced by reactive forces at A and B, in accordance with Newton's third law. Additional freedoms may be precluded to model greater restraint by the environment. However, Figure 7.12(a) does illustrate the *minimal* number of constraints.

Figure 7.12(b) is a simplification of Figure 7.12(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 Figure 7.12(c), however, it becomes ineffective in constraining the infinitesimal rotational motion about point A because the rolling direction is normal to AB. The configuration of Figure 7.12(c) is called a *kinematic mechanism*, and will result in a singular modified stiffness matrix.

§7.6.2. Supporting Three Dimensional Bodies

Figure 7.13 illustrates the extension of the freedom-restraining concept to three dimensions. The minimal number of freedoms that have to be constrained is now *six* and many combinations are possible. In the example of Figure 7.13, all three degrees of freedom at point A have been fixed. This prevents all rigid body translations, and leaves three rotations to be taken care of. 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.

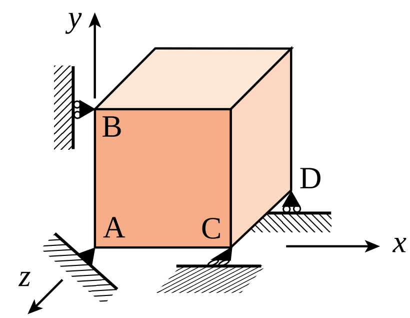


FIGURE 7.13. Suppressing rigid body motions in a three-dimensional body.

§7.7. 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.[9]

§7.7.1. Symmetry and Antisymmetry 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 7.14(a). Alternatively, a 180° rotation of the body about the symmetry line reproduces exactly the original problem.

An *antisymmetry line* can be recognized by the displacement pattern illustrated in Figure 7.14(b).

[9] Even if symmetry or antisymmetry are not explicitly applied through boundary conditions, they provide valuable checks on the computed solution.

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) as well as *rotational symmetry* can be treated in a similar manner, but will not be discussed here.

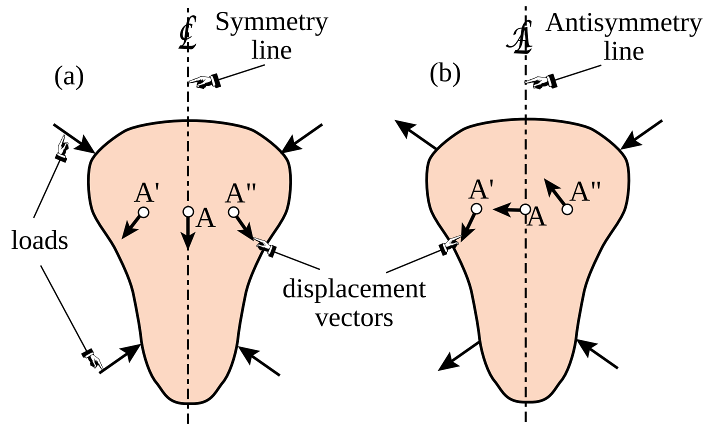


FIGURE 7.14. Visualizing symmetry and antisymmetry lines.

§7.7.2. 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.

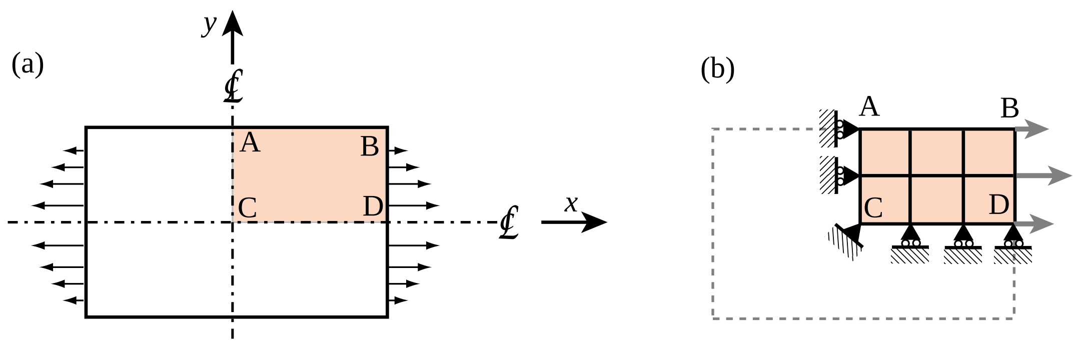


FIGURE 7.15. A doubly symmetric structure under symmetric loading.

Consider the plate structure shown in Figure 7.15(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 7.15(b).

On the other hand if the loading is *antisymmetric*, as illustrated in Figure 7.16(a), the x axis becomes an *antisymmetry line* as none of the $y = 0$ points can move along the x direction. The boundary conditions to be imposed on the FE model are also different, as shown in Figure 7.16(b).

Remark 7.3. For the case shown in Figure 7.16(b) note that all rollers slide in the same direction. Thus the vertical rigid body motion along y is not precluded. To do that, one node has to be constrained in the y direction. 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 7.16(b) the center point C has been so chosen. 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.

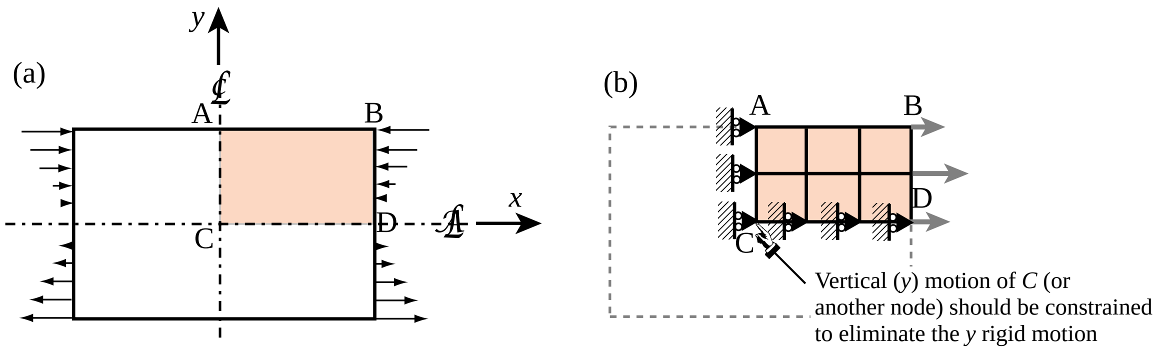


FIGURE 7.16. A doubly symmetric structure under antisymmetric loading.

Remark 7.4. Point loads acting at nodes located on symmetry or antisymmetry lines require special care. For example, consider the doubly symmetric plate structure of Figure 7.15 under the two point loads of magnitude P, as pictured in Figure 7.17(a). If the structure is broken down into 4 quadrants as in Figure 7.17(b), P must be halved as indicated in Figure 7.17(c). The same idea applies to point loads on antisymmetry lines, but there the process is trickier, as illustrated in Figure 7.18. The load must not be applied if the node is fixed against motion, since then the node force will appear as a reaction.

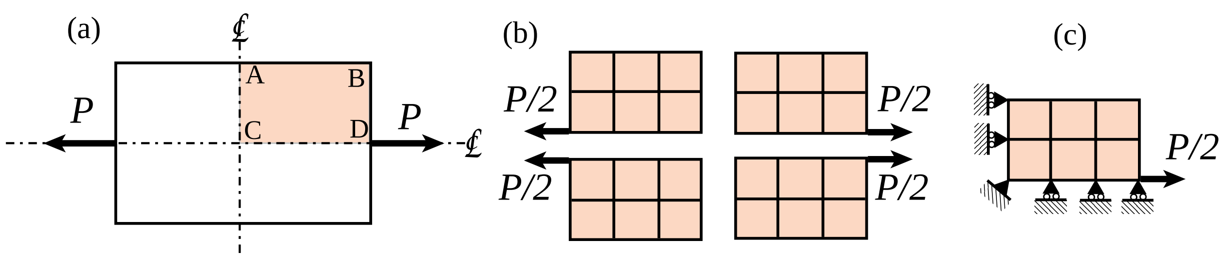


FIGURE 7.17. Breaking up a point load acting on a symmetry line node.

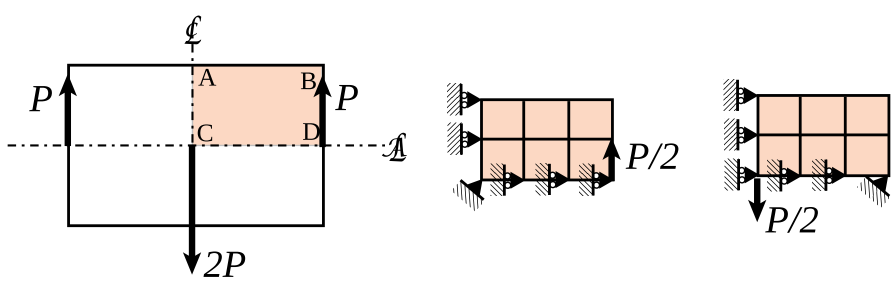


FIGURE 7.18. Breaking up a point load acting on a antisymmetry line node.

Distributed loads should not be divided when the structure is broken down into pieces, since the lumping process will take care of the necessary apportionment to nodes.

Notes and Bibliography

FEM modeling rules in most textbooks are often diffuse, if given at all. In those that focus on the mathematical interpretation of FEM they are altogether lacking; the emphasis being on academic boundary value problems. The rule collection at the start of this Chapter attempts to collect important recommendations in one place.

The treatment of boundary conditions, particularly symmetry and antisymmetry, tends to be also flaky. A notable exception is [437], which is understandable since Irons worked in industry (at Rolls-Royce Aerospace Division) before moving to academia.

The Norfork Dam used in Examples 7.1 and 7.2 (and two Exercises) for hydrostatic load-lumping calculations was the first realistic Civil Engineering structure analyzed by FEM. It greatly contributed to the acceptance of the method beyond the aerospace industry where it had originated. How this seminal event came to pass is narrated by Wilson in [883], from which the following fragment is taken. Annotations are inserted in squared brackets.

> "On the recommendation from Dr. Roy Carlson, a consultant to the Little Rock District of the Corps of Engineers, Clough [then a Professor at UC Berkeley] submitted [in 1960] a proposal to perform a finite element analysis of Norfork Dam, a gravity dam that had a temperature induced vertical crack near the center of the section. The proposal contained a coarse mesh solution that was produced by the new program [a matrix code developed by E. L. Wilson, then a doctoral student under Clough's supervision; the mesh is that shown in Figure 7.7(d)] and clearly indicated the ability of the new method to model structures of arbitrary geometry with different orthotropic properties within the dam and foundation. The finite element proposal was accepted by the Corps over an analog computer proposal submitted by Professor Richard MacNeal of CalTech [who later directed the development of NASTRAN under a NASA contract in the late 1960s], which at that time was considered as the state-of-the-art method for solving such problems.
>
> The Norfork Dam project provided an opportunity to improve the numerical methods used within the program and to extend the finite element method to the nonlinear solution of the crack closing due to hydrostatic loading. Wilson and a new graduate student, Ian King, conducted the detailed analyses that were required by the study. The significant engineering results of the project indicated that the cracked dam was safe [since the crack would be closed as the reservoir was filled]."

References

Referenced items moved to Appendix R.

Homework Exercises for Chapters 6 and 7

FEM Modeling: Mesh, Loads and BCs

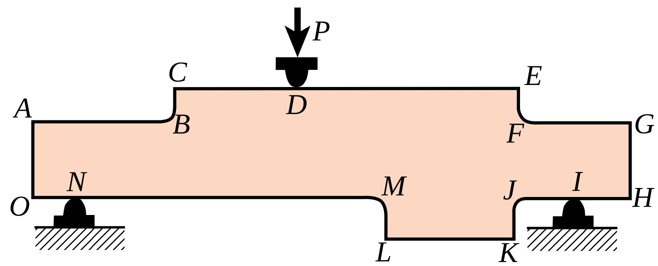


FIGURE E7.1. Inplane bent plate for Exercise 7.1.

EXERCISE 7.1 [D:10] The plate structure shown in Figure E7.1 is loaded and deforms in the plane of the paper. The applied load at D and the supports at I and N extend over a fairly narrow area. List 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.

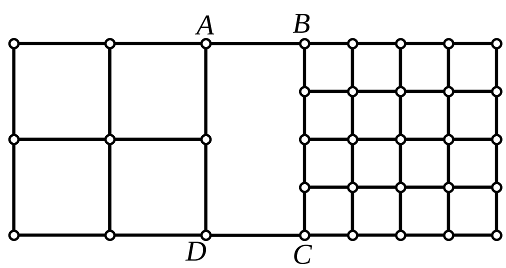


FIGURE E7.2. Transition meshing for Exercise 7.2.

EXERCISE 7.2 [D:15] Part of a two-dimensional FE mesh has been set up as indicated in Figure E7.2. 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.

EXERCISE 7.3 [A:15] A rectangular plate of constant thickness h and inplane dimensions $8a$ and $6a$ is meshed with 8 rectangular elements as shown in Figure E7.3(a). The plate specific weight is γ and acts along the $-y$ axis direction.

(a) Compute the node forces due to plate weight at nodes 1 through 15, using the NbN method. Obtain the node-tributary regions as sketched in Figure E7.3(b), which shows each element divided by the medians drawn as dashed lines (the tributary region of node 7 is shown in yellow). Partial answer: $f_{y1} = -2a^2\gamma h$. Check that adding up all y forces at the 15 nodes one gets $W = -48a^2\gamma h$.

(b) Repeat the computations using the EbE method. For this, take the total weight force on each element, and assign one quarter to each corner node. (This agrees with consistent energy lumping for 4-node rectangular elements.) Do the results agree with NbN lumping?

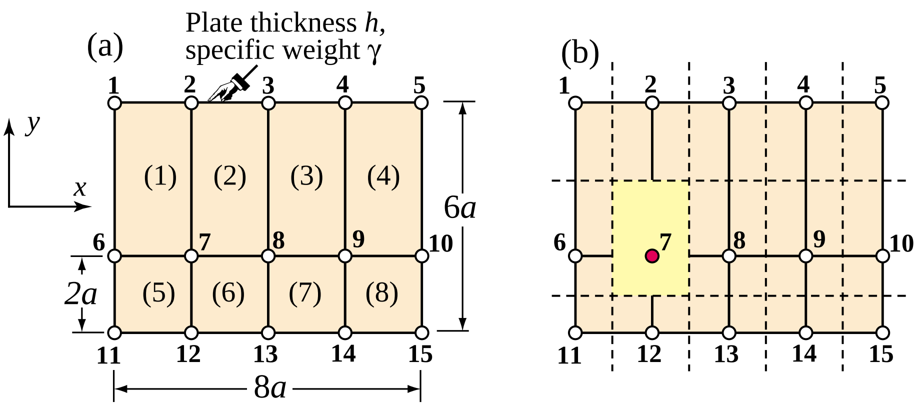


FIGURE E7.3. (a) Mesh layout for Exercise 7.3. (b) shows tributary area for node 7.

EXERCISE 7.4 [N/C:20] Complete the computation of hydrostatic node forces on the Norfork Dam under a water head of 180 ft, initiated in Example 7.1, using the data of Figure 7.8, and either the NbN or EbE method (pick one). Assume face AB is vertical. Do two checks: sum of horizontal (x) forces on nodes 1 through 5 is $\frac{1}{2}180^2 \times 62.4$ lbs, and sum of vertical (y) forces on nodes 5 through 9 is $-180 \times 62.4 \times 490$ lbs.

EXERCISE 7.5 [N/C:25] Complete the computation of own weight nodal forces of the coarse mesh of the Norfork dam proper, initiated in Example 7.2, reusing the data of Figure 7.9. Use the EbE method. Recover coordinates, and write a computer program to compute node forces. Compute the total weight of the dam slice in lbs.

EXERCISE 7.6 [A:10] Figure E7.4 depicts two instances of a pull test. In (a) a stiffer material (steel rod) is pulled out of a softer one (concrete block); in this case the load transfer diagram shows a rapid variation near the inner end of the rod. In (b) a softer material (plastic rod) is pulled out of a stiffer one (concrete rod), and the load transfer diagram is reversed.

If the test is to be simulated by a finite element model, indicate for (a) and (b) where a finer mesh would be desirable. Explain.

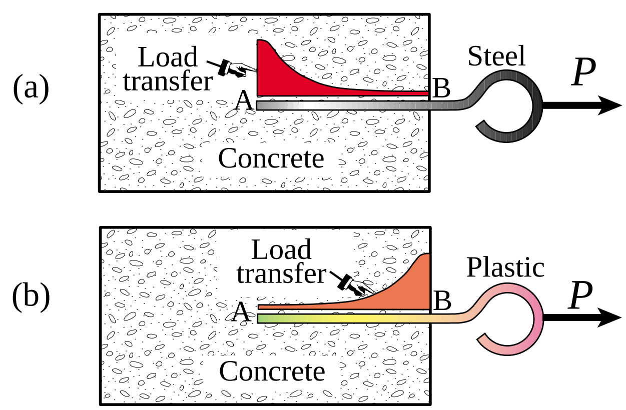


FIGURE E7.4. Pull tests for Exercise 7.6.

EXERCISE 7.7 [D:20] Identify the symmetry and antisymmetry lines in the two-dimensional problems illustrated in Figure E7.5. 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 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.[10]

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

[10] 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.

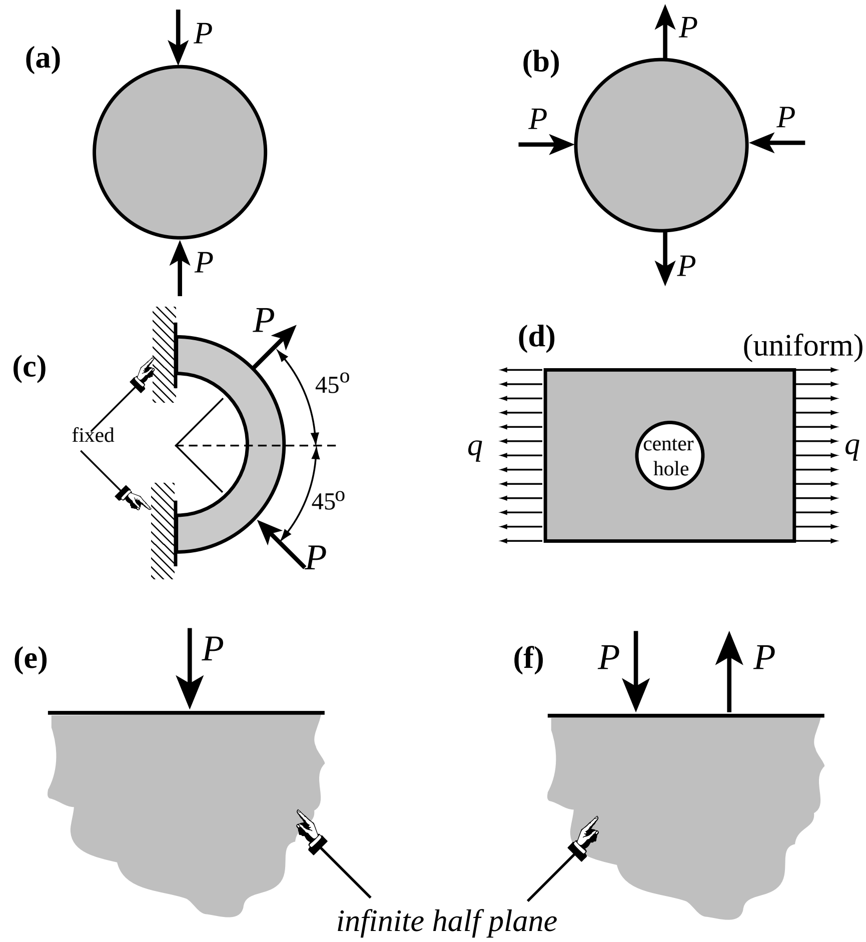


FIGURE E7.5. Problems for Exercise 7.7.

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 7.8 [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 E7.6.

After a long struggle, however, the worm wins. While hungrily looking for a smaller one your thoughts wonder to FEM and how the worm 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.

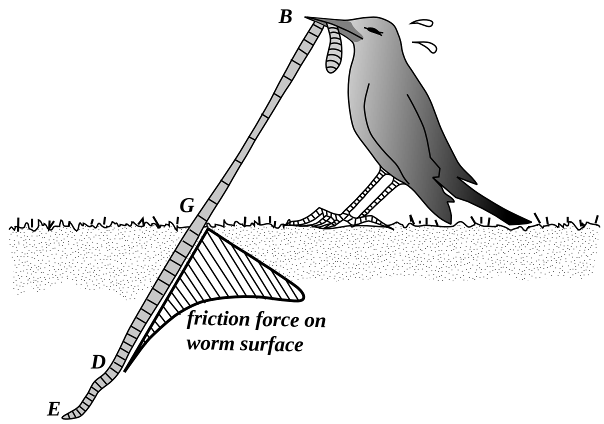


FIGURE E7.6. The hungry bird.

Briefly explain your decisions. Dont write equations.

EXERCISE 7.9 [D:15, 5 each] Prove from kinematics:

(a) Two symmetry lines in 2D cannot cross at a finite point, unless that point is fixed.

(b) Two antisymmetry lines in 2D cannot cross at a finite point, unless that point is fixed.

(c) A symmetry line and an antisymmetry line must cross at right angles, unless the cross point is fixed.

Note: proofs of (a,b,c) are very similar; just draw vectors at alleged intersections.

EXERCISE 7.10 [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 point C is fixed.

EXERCISE 7.11 [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 7.12 [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 7.13 [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 7.14 [A:20] Prove that EbE and energy consistent lumping agree if the element shape functions are piecewise linear.

EXERCISE 7.15 [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)