1 Finite Element Concepts

CHAPTER ROADMAP

In this chapter, we provide a simple introduction to the finite element method (FEM) and how it is related to other solution methods for engineering and physics problems. Throughout this chapter and the next, we avoid rigorous mathematical developments and equations and use simple examples that are easily understood by all students. We identify five basic steps or stages for any type of finite element analysis. These are: modeling and discretization; formulation and element equations; assembly; boundary conditions and solution; and finally postprocessing. In commercial FE programs these steps are lumped into three stages: modeling; solution; and postprocessing. One of the five steps, namely the assembly process, is quite simple and straightforward. A student may actually write a simple and general program in just one page that will do the assembly process. On the other hand, most of the research done and the textbooks written in the finite element area involve one or more of the other basic steps or stages. Throughout the introduction of the basic steps of the FE method, we introduce definitions of conceptual terminology that are common in the FE field, e.g., elements, nodes, boundary conditions, degrees of freedom (DOFs). To enable the students to start the modeling step we highlight the common elements used in most commercial programs and their geometry, nodes and DOFs.

This chapter, then, provides a brief account of the history of the development of the FE method. This is presented in two parts: the history of the development of the formulation and algorithms of the method; and the history of the development of computer hardware and software related to the application of the method. The final section of the chapter presents some typical applications of the FEM, mostly from the work of the author. These are meant to give students an overview of the capabilities and limitations of the method in various fields.

1.1 General Solution of Continuum Problems

There are several approaches to solving engineering problems. The diagram shown in Figure 1.1 summarizes the general solution procedures of continuum problems. Conventionally, continuum problems are solved by analytical and exact solution methods that are normally obtained by direct integration of the governing differential equations. However, analytical solutions are, generally, limited to simple geometries, simple boundary conditions and linear material models. Such conditions are, generally, not applicable to practical engineerng and design problems and therefore analytical solutions are usually difficult or even impossible to obtain.

To overcome the difficulties of analytical methods, numerical methods are employed to provide approximate solutions. Several numerical methods have been developed

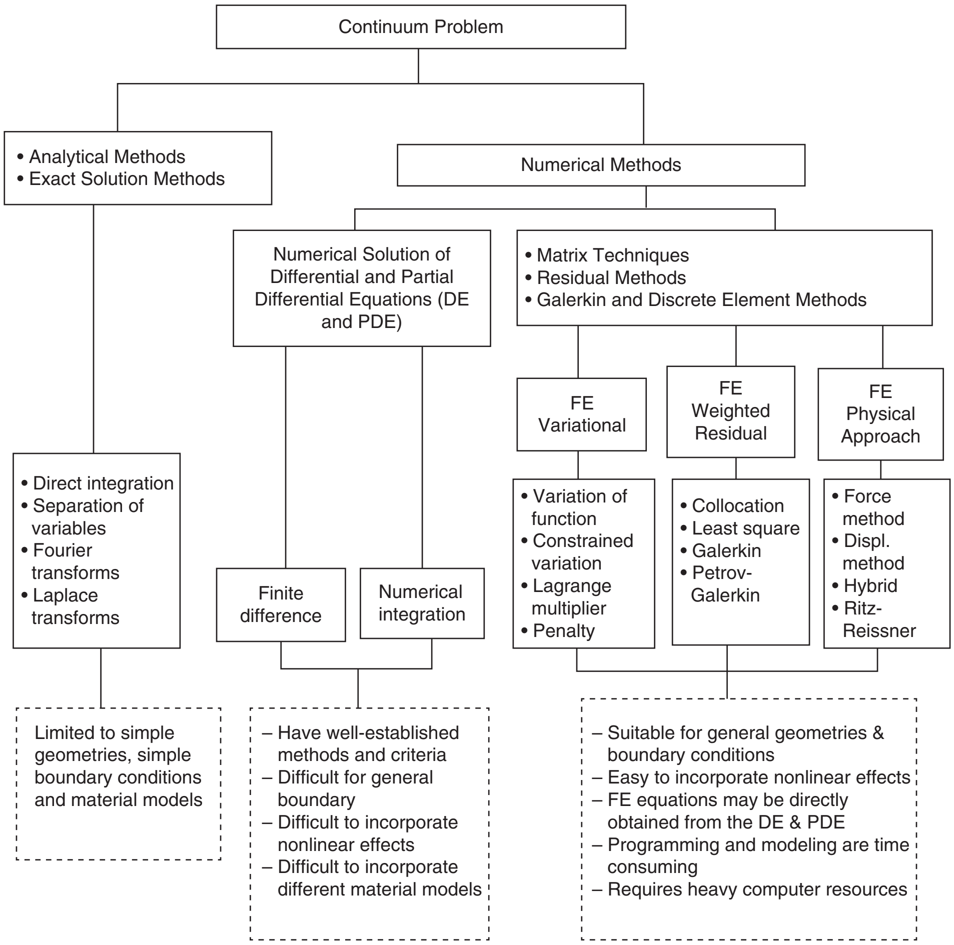

Figure 1.1 General solutions of continuum problems

over the past decades. Finite difference and numerical integration are two commonly used numerical techniques in which the governing differential or partial differential equations are solved numerically rather than in a closed form solution. Using these techniques, some fairly difficult problems have been solved. Over the past few decades, these numerical techniques have provided easier alternative solution procedures for problems involving different-material models, irregular geometrical boundaries, material nonlinearities (e.g., plasticity) and nonlinear boundary conditions.

Another, more recent, numerical approach is the finite element method, which is broadly defined as a group of numerical methods for approximating the governing equations of any continuous system. The finite element approach has a number of distinctive features which make it superior to other numerical approaches. It is suitable and easily adaptable for solving problems with general geometry and boundary conditions. Also, using this method it is straightforward to incorporate nonlinear effects and different-material models and the method may be applied to a wide range of engineering and physics problems. However, developing general purpose finite element programs requires intensive effort and many person-years of development. Additionally, building finite element models and interpreting the results of a model generally consumes a considerable amount of engineering time. Depending on the size of the model, heavy computer resources may be required.

Normally, all solution approaches start by analyzing and studying a very small part or domain of the structure, which we call an *element*. The underlying difference between the finite element approach and other numerical and analytical approaches is the choice of element size (refer to the diagram in Figure 1.2). In traditional analytical methods, an infinitesimal domain or element of the overall structure is used. The infinitely small element has infinitesimal dimensions, e.g., dx,

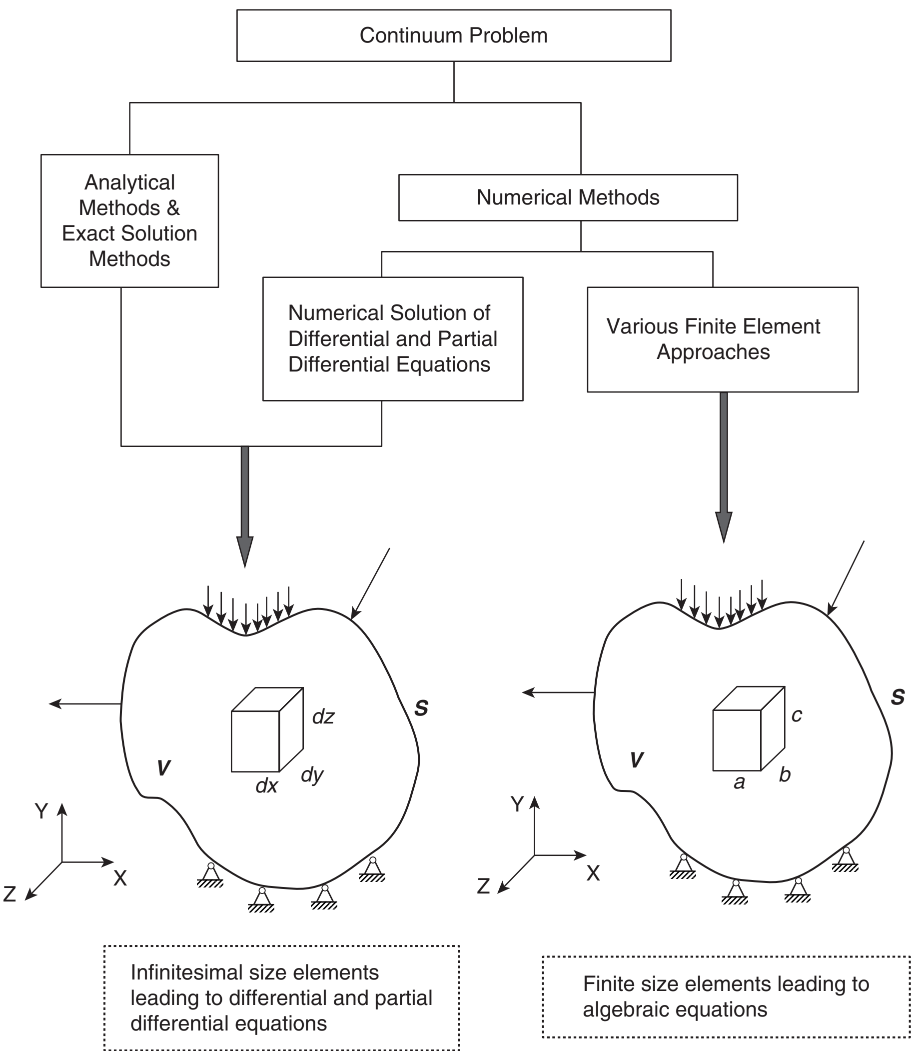

Figure 1.2 Differential and finite element methods

dy, dz in the case of three-dimensional analyses. The element equations for this domain are easily established from physical laws and by using simple calculus of variations. Since the dimensions of the domain or the element are infinitesimal, the resulting element equations will be differential or partial differential equations. In order to obtain the overall system behavior, the element equations have to be integrated over the entire domain of the structure and the boundary conditions need to be applied. This is where difficulties arise. The analytical or closed form integration process is only viable for simple geometries and boundary conditions. Incorporating nonlinear effects or nonlinear boundary conditions with complicated geometries normally leads to serious difficulties in integrating the element equations. In other words, the process may be viewed as dividing the physical system or structure into an infinite number of infinitesimal size elements and the governing differential or partial differential equations are integrated over the entire system.

On the other hand, in the finite element method, the physical system is divided or discretized into a number of small but finite-size elements with finite dimensions, e.g., *a, b, c* in three-dimensional analyses. The main difference here is that the element has small but finite dimensions and not infinitesimal dimensions. Because of this difference, certain approximations have to be assumed for the behavior of physical quantities over the domain of the element. Since the element is generally small, approximating the physical behavior within the element should be an easy task. This fact leads to a set of governing equations for the element that is algebraic. These element equations may be linear or nonlinear but they are still algebraic and not differential. To obtain the overall system behavior, the governing equations of all elements in the system are assembled to obtain a global system of equations. This process is analogous to the integration process in the analytical approach. Boundary conditions are then applied to the global system of equations for the overall structure. The global system of equations in the finite element approach may be linear or nonlinear algebraic equations. The solution of these equations may be obtained using matrix and linear algebra techniques.

Remarks

- *Approaches to solving engineering and physics problems usually start by analyzing a very small part or domain of the structure. The underlying difference between the finite element approach and other numerical and analytical approaches is the choice of the element size (refer to the diagram in Figure 1.2). In traditional analytical methods, an infinitesimal domain or element of the overall structure is used, leading to a differential equation or partial differential equation (PDE). In the finite element approach, a small but finite size of domain or element is used, leading to algebraic equations.*

Remarks (cont.)

- *The above remark indicates that, conceptually, if a problem can be described or solved by DEs or PDEs then the same would be true if finite element equations are used.*

1.2 What is the Finite Element Method?

In the above section, we presented a brief overview of the finite element method and its relation to and differences from traditional analytical methods. In this section, we provide a more detailed account of the basic ideas and procedures on which the method is based.

To explain the basis of the method, we consider a very simple example. Suppose we want to determine the area of an irregular-shaped geometry as shown in Figure 1.3. One obvious way to do this is to divide the area into basic geometries such as rectangles and triangles, to calculate the area of each subdivision, and then to sum these areas to get the overall required area of the irregular geometry. This very simple procedure essentially provides the basic steps of a generalized finite element procedure.

In this example, the area is divided into five rectangles and a triangle. In a typical finite element procedure, these subdivisions or subdomains are called *elements* and the process of subdividing the structure into small elements is called *discretization*, or *meshing*. Here, we may simply calculate the area of each of the subdivisions or elements. This is possible since we know the behavior of each subdivision or element, i.e., we know the element area equations. In the general case, however, the behavior and the equations of each subdivision or element may be unknown and have to be derived.

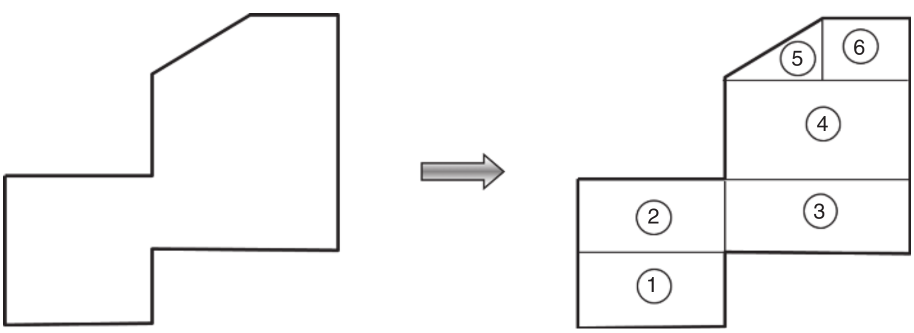

Figure 1.3 Area of an irregular shape

After establishing all element equations, we establish the global equations of the system. In our simple example, we seek to calculate the area of the irregular shape, which in more general terms represents the overall system behavior. In our example, this is simply achieved by summing all the areas of the individual elements to obtain the global picture or equation for the area of the structure. In a typical finite element procedure, this step is called the *assembly* and *solution*.

To summarize the above steps for our simple example: we first discretize the structure (called *meshing or modeling*), write the equations for each element (the *element characteristic equations*), add up all the individual element equations (*assembly*) and find the overall behavior of the structure (the *solution*).

To further clarify the above steps, we consider the example of calculating the area of a circle. We will assume that we do not know the equation for the area, i.e., the system behavior is unknown. First, we subdivide or discretize the circle into smaller elements that are easier to analyze, i.e., for which it is easier to calculate the area. In this case, we choose an element with a triangular shape and consider eight elements, as shown in Figure 1.4. The collection of the eight elements that we use for discretization is called the *mesh* or *model*. Unlike in the previous example, we realize that there is a slight difference between the area of the discretized model and the area of the actual problem or system. This difference is due to the fact that the chosen elements have straight edges that will not exactly match the boundary of the original system unless there is an excessively large or infinite number of elements. This leads to an important conclusion: the more elements we use, the more accurate the results will be. In fact this is not only due to the fact that the elements are straight sided. To explain this point further, we consider a more general case of discretization, in which we do not know the element equation or element behavior. In such a case, we have to introduce certain approximations to derive characteristic equations for the elements. Obviously, if we choose smaller element sizes, it will be easier to introduce more reasonable assumptions and the resulting element characteristic equations will be more accurate. Therefore, in order to increase the accuracy of the results, the densities of the elements have to be increased.

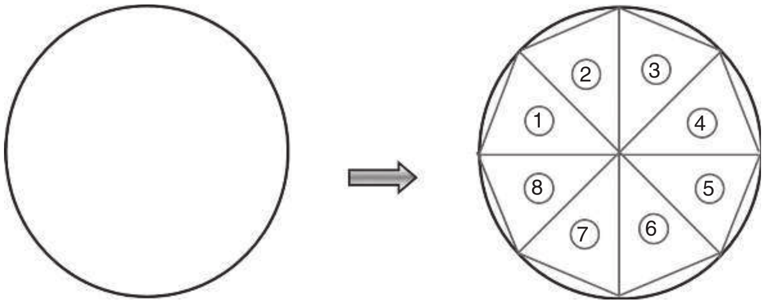

Figure 1.4 Area of a circle

The remaining steps are quite similar to those performed in the example corresponding to Figure 1.3. For this example, the element characteristic equations are those for the areas of each element. The next step is to sum all the element equations or to perform the assembly process. The global system of equations, or the area of the circle, will simply be the summation of all the element areas.

The above example is, to some extent, trivial. To enforce ideas, we now consider a slightly different and more elaborate example. Consider a beam or shaft problem, as shown in Figure 1.5. The beam has three different cross sections, A_1, A_2 and A_3 with corresponding lengths l_1, l_2 and l_3. The beam is subjected to an axial load P as shown in Figure 1.5 and it is required to calculate the deflection at the end of the beam.

Following the same steps as the previous two examples, we start by modeling the beam using simple elements. In this example, we choose three axial spring elements to discretize and model the beam. The element numbers in Figure 1.5 are identified by a circle around the number. The next step is to derive the element characteristic equations. In this case, we need to derive an equation that relates the force in each element to the response or the displacement of the element. This may not be a trivial equation, as in the previous two examples, but it may still be easy to derive. Each element is represented by an axial spring having only extensional degrees of freedom (DOF). The stiffness of each spring may be calculated using the equation

$$k_i = \frac{E_i A_i}{l_i} \quad (\text{no sum on } i) \tag{1.1}$$

where i is the element number, k is the element stiffness, E is Young's modulus for the element material, A is the cross sectional area of the element and l is the element length. Having calculated the stiffness, the net extension or net displacement in each element may be readily obtained from the following equation:

$$F^{(i)} = k_i\, u \tag{1.2}$$

where $F^{(i)}$ is the force on the spring element, k_i is the element stiffness and u is the net extension or displacement in the element. Although Equation 1.2

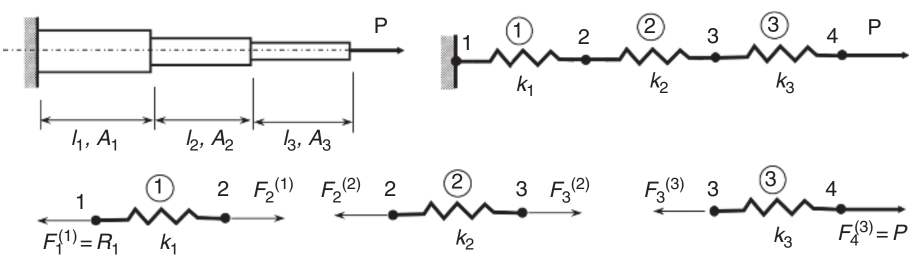

Figure 1.5 Extension of a beam or shaft

seems simple, it requires further discussion. We start by looking at the force on the element, $F^{(i)}$. Since Equation 1.2 represents the behavior or the equilibrium equation of the element, the element forces will be those forces acting on the element when it is separated from the structure as a free body. These are the reactions from the structure on the element at the points where the element is connected to the structure or what is normally called the *element internal forces*. Obviously, in the case of a simple spring element, these will be equal and opposite forces at the end points of the element. Thus, for element 1, we have two equal and opposite forces at points 1 and 2 $(F_1^{(1)}, F_2^{(1)})$; for element 2, we have two equal and opposite forces at points 2 and 3 $(F_2^{(2)}, F_3^{(2)})$; and for element 3, we have two equal and opposite forces at points 3 and 4 $(F_3^{(3)}, F_4^{(3)})$. The free body diagram for each element is shown in Figure 1.5. An important note should be given here. The element forces have been identified with a superscript indicating the element number and a subscript indicating the point of application. This is now essential, since, in general, we may have more than one element meeting at a given point. The equilibrium at each such point indicates that the sum of the external and internal forces at the point must be zero and the external forces at such a point will be shared by the elements meeting at the point. These points of connection between the element and its neighboring elements or with the supports are called *nodes*. In this example, we have four nodes, marked as 1, 2, 3 and 4. The equilibrium of each node requires that the vector sum of all the forces at this node be zero, in the case of free nodes such as nodes 2 and 3, and equal to the external force or the reaction at nodes 4 and 1. According to the assumed force directions in Figure 1.5, this leads to the following relations for the nodal forces:

$$F_1^{(1)} = R_1, \quad F_2^{(1)} = -F_2^{(2)}, \quad F_3^{(2)} = -F_3^{(3)}, \quad F_4^{(3)} = P \tag{1.3}$$

Now, we turn our attention to the response or the displacement of the element. In this case, the response is simply measured by the net displacement in the element. This will be $u_2 - u_1$ for element 1, $u_3 - u_2$ for element 2 and $u_4 - u_3$ for element 3.

Equation (1.2) for each element may now be rewritten in a more detailed way as follows:

$$\begin{aligned}
&\textit{For element } 1: \quad && F_2^{(1)} = k_1(u_2 - u_1), && \text{and} \quad && F_1^{(1)} = R_1 = -k_1(u_2 - u_1),\\
&\textit{For element } 2: \quad && F_3^{(2)} = k_2(u_3 - u_2), && \text{and} \quad && F_2^{(2)} = -k_2(u_3 - u_2),\\
&\textit{For element } 3: \quad && F_4^{(3)} = P = k_3(u_4 - u_3), && \text{and} \quad && F_3^{(3)} = -k_3(u_4 - u_3)
\end{aligned} \tag{1.4}$$

It should be noted that in Equations 1.3, 1.4 we have assumed that the forces on each element are equal in magnitude and opposite in direction.

The next step is to assemble all element equations to obtain the overall system or global equations. By eliminating u_2 and u_3 from equation 1.4 and noting that all the forces in the elements are simply equal to P, we may obtain the following relation for the response of the system:

$$u_4 = P\left(\frac{1}{k_1}+\frac{1}{k_2}+\frac{1}{k_3}\right)+u_1 \tag{1.5}$$

The above equation represents the overall system behavior before the boundary conditions are applied. It should be noted that by assigning any value to u_1 we have infinite solutions for the displacement u_4. In order to complete the solution for the response of the system, i.e., in order to obtain a unique value for the displacement u_4, we need to introduce the boundary condition which is simply $u_1 = 0$. It should be noted here that without introducing boundary conditions, the global system equations (Equation 1.5 in this case) will have infinite solutions. For the general case, this means that the matrix representing the global system of equations is singular and is not a positive definite matrix.

Introducing the boundary condition $u_1 = 0$ reduces the final system or global equation to the following form:

$$P = K_g u_4 \quad \text{where} \quad K_g = 1/\left(\frac{1}{k_1}+\frac{1}{k_2}+\frac{1}{k_3}\right) = 1/\sum_{i=1}^{3}\left(\frac{1}{k_i}\right) \tag{1.6}$$

Equation 1.6 represents the assembly process and may be readily verified by summing the stiffness of the three spring elements. The final solution step is to solve Equation 1.6 to obtain the displacement u_4.

Although the above example is a simple one, it essentially introduces the basic steps involved in linear finite element structural analysis, which may be summarized as follows:

1. **Modeling and Discretization (Preprocessing):**
   In this step, the structure is divided into small elements that will be easier to study.

2. **Element Characteristic Equations:**
   In this step, each element is studied separately to derive its equilibrium or characteristic element equations.

3. **Assembling of Element Equations:**
   In this step, all element equations are assembled into a global system of equations. In the third example presented above, this process is performed by eliminating internal variables. This is not the general approach, and a more systematic method will be presented to establish the assembled global equations of the system. In general, this step is straightforward and simple.

4. **Solution of Global Equations:**
   In this step, we first apply the boundary conditions of the global structure. Without applying the proper boundary conditions of the structure, the global equations may have an infinite number of solutions and the matrix representing the global equations will be singular.

We now present another example to clarify and highlight the basic procedures discussed in the above section.

EXAMPLE 1.1

It is required to calculate the area of the irregular shape given in Figure 1.6. Also, it is required to calculate the ratio of the bottom leg's rectangular area to the total area of the structure.

Approach and Assumptions

This is a simple example in which we demonstrate the solution steps and which we will use as a reference for other more realistic examples. Since we do not know an equation to give us the area of the shape, we divide the shape into smaller regions (later called *elements*) that we can easily handle (i.e., in this case, whose area we can easily find). There is no conceptual constraint that requires these elements to be of the same type and/or shape.

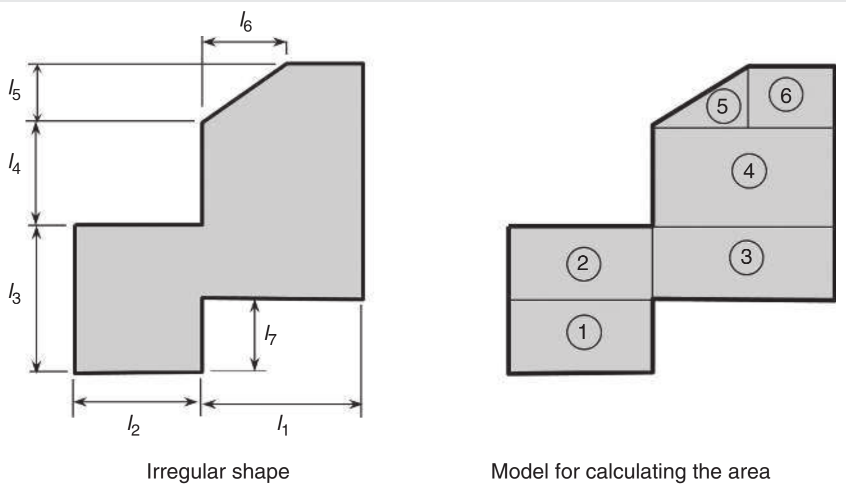

Figure 1.6 Calculating the area of an irregular shape