CHAPTER 2

Structures modeling using Finite Elements

2.1 Introduction

Each time a model is constructed an abstraction from reality is created; in other words the aim is to get the problem back to simplified and well known schemes, which can give an interpretation of the reality itself. For example Solid Mechanics Theory gives us the analytical solution for the behaviour of beams subjected to loads.

Nevertheless we know that the De Saint Venant relations are valid under at least one restrictive hypothesis: the beam has to be a one-dimensional body, i.e. the two transversal dimensions have to be negligible with respect to the axial one. And this represents just a model of the reality, a simplification that gives good and valid results in many technical cases.

However frequently this method is abused, extending arbitrarily the validity of this model over its intrinsic limits, thus violating the hypothesis under which the model was originally built.

In order to go over this and other boundaries, avoiding to arrive to unaffordable calculation results, it has been necessary to develop a general validity method, that could experience in a reduced way the limitations posed by too restrictive hypothesis, bounded to peculiar cases.

It is clear that the ideal method would be the one that could solve in analytical form the mixed system of algebraic and partial differential equations shown in Chapter 1 (see equations (1.3) and (1.8)). In spite of this, as we already said, the analytical solution presents difficulties that are practically insurmountable, a part from particular cases that, being so, make vane the effort to gain a general approach.

So here is the idea to develop a method that could solve the system at least in a given domain, limited in the space and with a simple shape: in this way, by dividing the original domain of interest, which is surely bigger and with a geometry more articulated, in an adequate number of simple subdomains for which the solution is known, it is possible to achieve the solution of the original problem by "reassembling" the partial results opportunely.

By following this approach, the problem has been discretised and the solution is certainly an approximation of the reality, but in engineering practical cases this result is satisfactory.

The mentioned subdomains, in which the domain is discretised, are known as Finite Elements.

Nevertheless even solving the elastic problem in a single element is not as simple. In order to proceed it is necessary to introduce a further approximation which imposes that the displacement of a point, located at the internal of an generic element, is a function (linear, parabolic, bilinear, etc. depending on the element characteristics) of the displacements of predefined points (known as grids) of the element itself. These relations are known as Shape Functions, and they are equations that govern the displacements of all the points belonging to an element, depending on how the grids of the element itself move in the space (in Appendix B it is possible to find out further details about Shape Functions, at least for plane stress, and on the use that Finite Element Method makes of them).

From what we said, and from what reported in Chapter 1, it is clear that from the knowledge of the displacement components of the grids, that connect all the elements in which the structure has been divided, it is possible to identify the strain status and thus the stress status of the structure. Finite Element Method (FEM) is based on the "displacement method", which is treated in Solid Mechanics Theory in order to solve over constrained structures; the only result obtained from a FEM code, having solved the equations, is the displacements field of the grids: all the other quantities (i.e. strains, stresses, etc.) are derived from here.

So, in order to analyse a generic structure by means of the Finite Element Method, it is necessary to follow some points, briefly indicated here below:

- choice of the element type to be used, depending on the structure geometry and on the phenomenon to be investigated
- division of the structure in an "adequate" elements number
- applications of the boundary conditions (constraints and loads)
- solution of the equations
- results interpretation

Each one of these phases, which are in different measures user dependent, represents a critical task and can invalidate all the others.

For example a mesh with bad "numerical quality" will surely give to the equations solver algorithm some difficulties, as we will see in Chapter 5, thus not allowing to obtain good results. Furthermore a "perfect" model can be solved by a not robust solution algorithm, generating poor quality results. Moreover the not perfect knowledge of the physics at the base of the problem can lead to the construction of a poor model (mistakes in choosing the element type, erroneous applications of boundary conditions, inadequate material description, etc.)

Last but not least, results interpretation is the most critical phase, because it needs a good knowledge from the stress engineer. In fact it has always to be kept in mind that computers, and all the codes implemented on them, are only instruments that can handle and manage lots of data, equations and numbers. Only the engineering judgment of the user can validate the results obtained by a calculation.