CHAPTER 5

Errors in Finite Element calculation

5.1 Introduction

Up to now we have seen how to "handle" a finite element model and how to manage the results that it produces. Nevertheless these numbers are greatly influenced by the way the model is built and by the numerical quality with which the equations generated by the model are solved.

It is clear that the "trash in-trash out" principle is valid; in other words if the input is affected by errors it is not possible to expect a correct output.

The errors that can affect a numerical analysis are different: starting from errors due to the user (i.e. an incorrect material description) or errors introduced by the software used for the model construction and arriving to errors implicit in the method (we remember that FEM approximates the solution) or numerical errors.

In the next paragraphs of this Chapter we will deal with all these kinds of errors, highlighting all the dangers, wherever possible, by means of simple examples.

5.2 User errors

This kind of error is probably the most difficult to deal with because it has its basis on the peculiarity of each person and it is therefore not predictable.

Nevertheless some classical cases do exist, related to the fact that it is necessary to input some numbers in the model; in fact today it is usual to work with graphical systems and the "keyboard inputs" are quite a few. It sometimes happens that the user makes a mistake in typing some numbers; as an example it is possible to type an incorrect value of the Young's modulus for the material used for the structure under analysis; it is possible to input wrong values of forces/pressures, of temperatures, etc.

A more conceptual mistake, still related to a numerical input, is the one based on the unit system that has to be used: calculation codes should all be "unitless", meaning that the user should be responsible in using a coherent unit system. If for example the International System (I.S.) is used, it is impossible to make errors, but the linear dimensions have to be input in meters and this could be sometimes (mainly in a mechanical environment) uncomfortable when "small" numbers are used (let us think of the nodal coordinates). Very often "hybrid" systems are used in order to manage easily the model geometry, but attention must be paid to the types of analysis that have to be conduced on a given model. A classical example is the use of the "modified" I.S.

where millimetres are used instead of meters; other quantities involved are forces (Newtons), masses (kilograms), time (seconds). For static analysis generally there are no problems: Young's modulus is expressed in MPa, as for pressures, and material density is input in kilograms on millimeters to cube. Apparently all works well, because the total mass (that the code calculates as the product of density multiplied by the total volume) gives a correct value. Nevertheless if the same model has to be used for dynamic analysis (without thinking of complicated things let us just take into account a natural frequencies analysis) we are going to obtain incorrect results (i.e. the numbers are "scaled" by a certain factor). Let us see the reason for that. Still thinking of modal analysis, the equation that gives the vibration frequency of a mass-spring system having M and K for the mass value and the spring stiffness respectively is:

$$\omega = \sqrt{\frac{K}{M}}$$

Using the I.S. system we would have that the dimensions of K and M would be respectively [N/m] e [kg]. Going down to base units we would have ($1\ N = \frac{1\ kg \cdot m}{s^2}$):

$$\omega = \sqrt{\frac{K}{M}} = \sqrt{\frac{\left[\frac{kg \cdot m}{s^2}\right] \cdot \left[\frac{1}{m}\right]}{[kg]}} = \left[\frac{1}{s}\right]$$

On the contrary, using the modified I.S. the dimensions of K are [N/mm] and therefore:

$$\omega = \sqrt{\frac{K}{M}} = \sqrt{\frac{\left[\frac{kg \cdot m}{s^2}\right] \cdot \left[\frac{1}{mm}\right]}{[kg]}} = \sqrt{\frac{\left[\frac{kg \cdot 1000 mm}{s^2}\right] \cdot \left[\frac{1}{mm}\right]}{[kg]}} = 10 \cdot \sqrt{10}\left[\frac{1}{s}\right]$$

It is evident that the calculated frequency is incorrect with respect to the actual one (in particular it is lower than the effective one). To "reorder" the things it is sufficient to express the masses in tons (i.e. the density in tons on millimeters to cube). Infact:

$$\omega = \sqrt{\frac{K}{M}} = \sqrt{\frac{\left[\frac{kg \cdot m}{s^2}\right] \cdot \left[\frac{1}{mm}\right]}{[t]}} = \sqrt{\frac{\left[\frac{kg \cdot 1000 mm}{s^2}\right] \cdot \left[\frac{1}{mm}\right]}{1000[kg]}} = \left[\frac{1}{s}\right]$$

In this case we have to remember that the total mass is expressed in tons instead of kilograms. Moreover if the model is subjected to inertial loads, even in static analysis, (i.e. the loads that come from masses and from accelerations) such as weight and centrifugal forces, the corresponding accelerations have to be expressed in millimeters over square second instead of meters over square second.

In order to try avoiding this kind of errors (that could be very serious) by the non export user, many new commercial codes have introduced inside them the use of a unit system. The user can now choose among different homogeneous unit systems. Very often it is possible to see some pictures where stresses are expressed in mMPa (that is, milli-mega Pascal: it is too easy to indicate kPa, kilo-Pascal!); and this because a method to avoid the above illustrated error is to express the forces in mN (milli-Newton).

Fortunately for the more expert ones, these codes have not yet taken the complete control over the user; it is possible to "force" them in using the numbers that each one prefers, thus leaving to the user responsibility the correctness of the choice.

Another "keyboard input" error is the erroneous input of the thickness to be assigned to shell or plane stress elements; in order to avoid these errors the only way is to check the model carefully.

Clearly the user can also make some "mouse input" errors in assigning forces, constraints, thermal loads, pressures to the wrong nodes or elements; also in these cases the main checking consists in a very careful analysis of the model, a part some tricks that we will see in Chapter 7.

5.3 Discretization errors

Discretization error is intrinsic to FEM. We said (and more details can be found in Appendix B) that FEM gives an approximate solution; the finer the discretization is, the more accurate the results will be. We highlight that here, with "discretization" we mean the way a structure is approximated: therefore we are thinking of the element type and the number of elements that we decide to use when we build the model of a given structure.

The number of elements is qualitatively referred to as "mesh density". It must be said that generally finite element modelling tends to overestimate the effective value of the structure stiffness; this is more and more true as the mesh is more and more coarse. This fact means that the stress values are underestimated; let us see the reason for this.

We well know that a bar loaded by an axial force F undergoes a deformation ε given by:

$$\varepsilon = \frac{\Delta L}{L}$$

being L the initial lenght of the bar; we also know that $\sigma = \varepsilon \cdot E$.