CHAPTER 1
Modeling of structures by Finite Elements

1.1 Introduction

Every time a model is created, an abstraction of reality takes place; that is to say, we try to lead back to known and simplified schemes that can give an interpretation of reality itself. For example, Solid Mechanics provides us with the analytical solution for the behavior of beams subjected to the action of loads. However we know that De Saint Venant's relations are valid under at least one restrictive hypothesis: the beam must be a one-dimensional body, in the sense that the two transversal dimensions must be negligible with respect to the axial length. And this represents just a model of reality, a simplification that provides very valid results in many technical cases.

However, it is often abused, arbitrarily extending the validity of this model beyond its inherent limits and violating the assumptions under which it was originally created.

In order to cross this and other boundaries, without risking to obtain unreliable calculation results, it was necessary to develop a method of general validity, which would suffer in a smaller degree from the limitations imposed by assumptions that are too restrictive and linked to particular cases.

It goes without saying that the ideal method would be the one that allows to solve in analytical form the mixed system of algebraic and partial differential equations that describe the elastic problem (and that we report synthetically in Appendix A). However, the analytical solution presents practically unsurmountable difficulties, except in particular cases that, as such, frustrate the attempt of generalization.

Here then comes the idea of developing a method that could solve the system at least in a domain limited in space and with a simple geometric shape: in this way, subdividing the domain of interest, certainly larger and with a more articulated geometry, in an appropriate number of simple subdomains for which the solution is known, it is possible to obtain the solution of the original problem "reassembling" adequately the partial results.

In this way the problem has been discretized and the solution obtained is certainly an approximation of reality, but in practical engineering cases this result is more than satisfactory.

The subdomains into which the source domain is discretized are called Finite Elements.

However, even solving the elastic problem within a single element is no small matter. In order to proceed it is necessary to introduce a further approximation and impose that the displacement of a generic point within the element is a function (linear, parabolic, bilinear, etc. in relation to the characteristics of the element) of the displacements of predefined points (called nodes) of the element itself. These relations are called Shape Functions. These are nothing more than equations that govern the displacement of all points within the element, in relation to how the nodes that belong to the element move (in Appendix B more details on the Shape Functions can be found, at least for the plane stress state, and on the use that the Finite Element Method makes of them).

From what has been said, and from what is reported in Appendix A, it is clear that from the knowledge of the displacement components of the nodes, which connect each other the elements in which the structure has been divided, it is possible to go back to the strain and stress states of the structure itself. The Finite Element Method (FEM) is therefore based on the "method of displacements", which is taught in Solid Mechanics courses for the resolution of statically indeterminate structures; the only result in output from a FEM calculation code, following the solution of the equations, is precisely the field of nodal displacements: all the other quantities are derived from here.

So, in order to analyze any given structure using the Finite Element Method, it is necessary to proceed through some points, briefly indicated below:

- identification of the type of element to be used, in relation to the geometry of the structure and to the phenomenon that is to be investigated
- subdivision of the structure into an "adequate" number of elements
- imposition of boundary conditions (constraints and loads)
- resolution of the equations that derive from the model
- interpretation of results

Each of these steps, more or less borne by the user, represents a criticality and can frustrate all the others.

For example, a mesh with elements of very poor "numerical quality" will certainly cause difficulties for the solving algorithm, as we will see in Chapter 6, and will compromise the validity of the results. Or a "perfect" model can be subjected to the solution by a mediocre code, generating results of poor quality. Or still the not perfect understanding of the physics that is at the base of the phenomena that are to be investigated can lead to the creation of an incorrect model (wrong type of element, wrong boundary conditions, inadequate description of the material, etc.).

The interpretation of the results is the phase that mostly lays its foundations on the preparation of the structural engineer. It must never be forgotten, in fact, that the computer, and all the calculation codes implemented on it, are only tools to manage and manipulate equations and numbers. Only the engineering judgment of the structural engineer can validate the results of a calculation.

1.2 Modeling with 2D elements

Clearly the best element is the one that can represent any stress and strain state, in its generality. However, there are conditions in which the problems can be reduced to simpler cases, without losing the accuracy of the results. In virtue of this fact special finite elements have been created (for more details see Appendix A).

In this section we will focus on the implementation of plane models, meaning that the equations governing them are those presented in § A.5, A.6, and A.7.

In general to resort to the simplification means to have smaller difficulties in the realization of the model (that translates in smaller possibilities of error), smaller times of calculation, more contained dimensions of the files. It is therefore always desirable to employ plane models, provided obviously that the simplification is lawful.

1.2.1 Plane stress

As it can be seen from § A.5, a plane stress state occurs when:

$$\sigma_{zz} = \tau_{yz} = \tau_{xz} = 0$$

A sufficient condition to have $\sigma_{zz} = 0$ is that the thickness (which extends in the z-direction) is "small" compared to the other two dimensions of the structure. This is the case, for example, with sheet metal.

A sufficient condition for having τ_{yz} and $\tau_{xz} = 0$ is that the thin structure in consideration is not loaded with shear forces normal to its surface. From all of the above, it is apparent that in order to model a structure with plane stress state elements, it is necessary that the forces that stress it belong to the same plane in which the structure lies.

A case of a plane stress state, for example, is represented by spur gears where the thickness is small relative to the other dimensions.

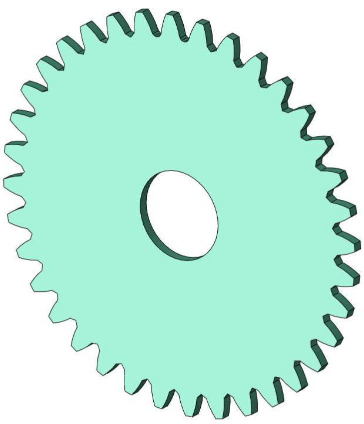

Figure 1.1. Spur gear. Thickness = 3mm, modulus = 3.2 mm, number of teeth = 36, pitch diameter = 115.2 mm.

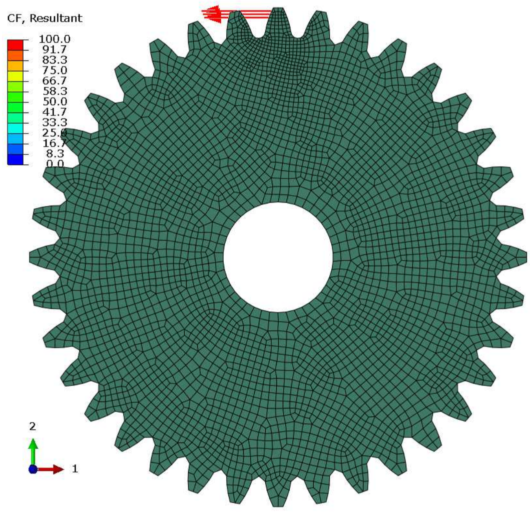

Figure 1.2. Finite element model in plane stress state for the gear in Figure 1.1. Note how the loaded tooth has been divided into more elements than the others.

Figure 1.1 shows the 3D CAD model of a gear satisfying the plane stress state assumptions; figure 1.2 illustrates the corresponding finite element model. Lastly, in figure 1.3 we depict the plotting of the equivalent Von Mises stress for the engaged tooth.

Here the equivalent Von Mises stress has been given, but it is possible to request any quantity, for example the maximum and minimum principal stresses, or the shear stress. It must be clear, however, that if one were to request the behavior of the stress tensor component normal to the gear surface (σ_{zz}), one would get a uniform coloring and a color scale full of zeros. We know, in fact, that we are dealing with a plane stress state.

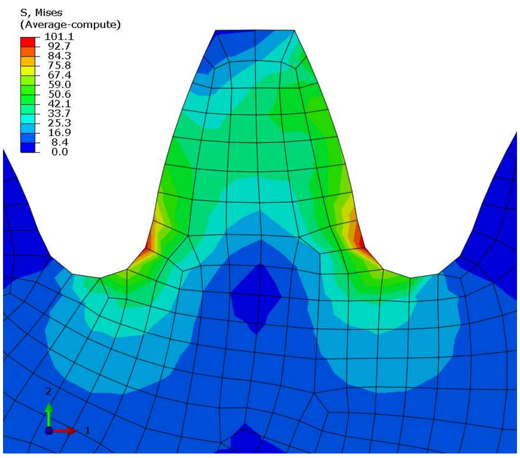

Figure 1.3. The results of the calculation. In this case, the equivalent Von Mises stress is represented (maximum value equal to 101 MPa).

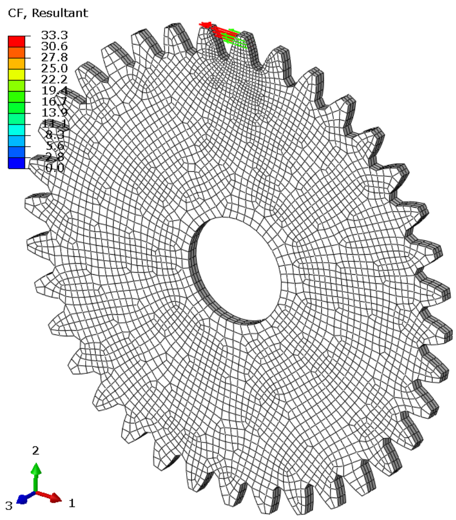

Figure 1.4. Solid finite element model for the gear in figure 1.1.

To carry out a comparison we build a model with solid elements (valid for modeling 3D structures), which allows us to reproduce the geometry of the part more faithfully. In this way we can compare the results we will obtain from the more refined but "heavier" model with those of the plane model.

Figure 1.4 illustrates the brick element model.3 elements were placed in the thickness, to try to capture the variation in tension σ_{zz} as well, which is not possible with the plane model.

Figure 1.5 shows the Von Mises equivalent stress contour for the 3D model, while figure 1.6 shows the stress σ_{zz}.

The differences in the results provided by the two models are less than 2%, an absolutely acceptable error in all cases of the technique. We then observe that the σ_{zz} stress appears to be decisively negligible compared to the other stresses involved. This comforts us on the validity of the results given by the plane stress model.

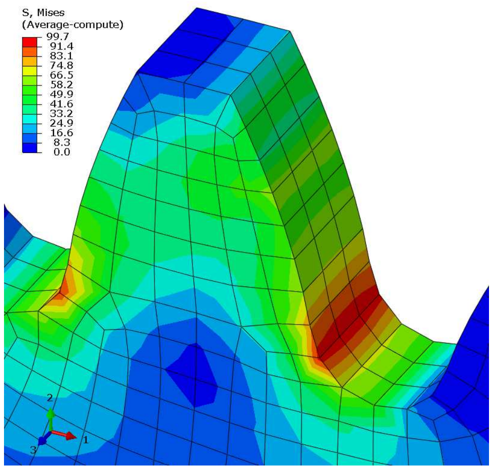

Figure 1.5. Von Mises equivalent stress (maximum value is 99.7 MPa).

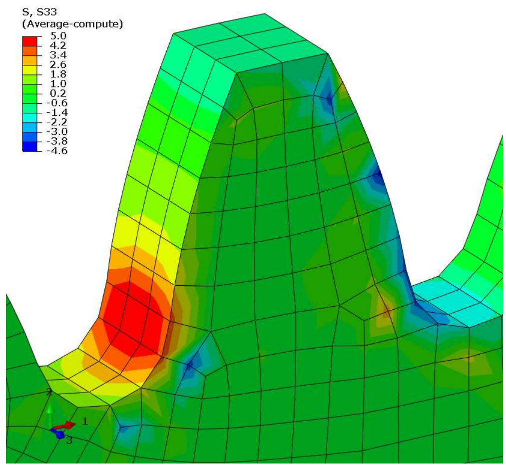

Figure 1.6. σ_{zz} stress (maximum value is 5.0 MPa).

<u>Remarks</u>

On a practical level, to realize a plane stress model means to indicate to the calculation code the type of element that is being used (and generally this operation is done by means of the graphic pre-processor); one must be careful because many finite element programs require that the element lie in one of the three planes of the global reference system (xy, yz or xz); clearly an incorrect positioning will produce an error message from the solver. Since the plane model has no physical thickness, this additional information must also be passed to the code.

The plane model results decisively more compact (303 KB for the input file, 1460 KB for the results file) in comparison to its "elder brother" 3D (1435 KB for the input file, 5804 KB for the results file); correspondingly also the times of calculation are more reduced. Even if today the power of the processors and the capacities of the mass memories do not represent anymore a limitation, simplifications reduce anyway the possibilities of error.

The mesh was refined at the loaded tooth; this allows the strain and stress gradients to be adequately captured, ensuring good results while reducing the number of equations. We will come back to the importance of mesh density in Chapter 6.

Looking at the geometry of the gear as a whole and thinking about how this type of organ works, it would have been possible to avoid modeling all the teeth, limiting the model to only the two or three teeth adjacent to the loaded one. This would have been another valid simplification.

The σ_{zz} of figure 1. 6, then, presents at the base of the tooth opposite signs: on the loaded side, in particular, it is negative, while from the opposite side it is positive; this fact should not be surprising since the tooth, basically, works in bending and therefore the loaded side will see the fibers stretched and the opposite side will see compression: it follows that from the stretched side the material will tend to contract (due to the Poisson coefficient), generating a negative σ_{zz}, while from the compressed side it will tend to expand, generating a positive σ_{zz}.

1.2.2 Plane strain

Dual to the plane strain state is the plane strain regime. As it can be seen in Appendix A, a strain state is plane when:

$$\varepsilon_{zz} = \gamma_{yz} = \gamma_{xz} = 0$$

In this case necessary condition to have $\varepsilon_{zz} = 0$ is that the dimension normal to the plane in which the structure lies is preponderant with respect to the others. For example, if the gear in figure 1.1 had a 15 mm thickness instead of 3 mm, we could already be talking about plane strain.

Similarly to what happens for the plane stress state to have $\gamma_{yz} = \gamma_{xz} = 0$ it is necessary that the structure is loaded with forces that lie in the plane to which the structure belongs.

The equations governing the plane strain regime are those reported in § A.6. We observe that in this case the component σ_{zz} of the stress tensor is not zero, due to the transverse contraction coefficient (see eq. A.13).

As an example in this case we will calculate the stress state arising inside a thick and long cylinder, in a section away from flanges or edges that may alter the load path, when a pressure p = 50 MPa acts inside it.

The stress in the tangential direction (σ_t), according to the Lamé formulas, is worth:

$$\sigma_t = p \cdot \frac{\left(\frac{D_e}{d}\right)^2 + 1}{a^2 - 1}$$

being D_e the external diameter, D_i the internal diameter interno, d the generic diameter and $a = \frac{D_e}{D_i}$.

Let us assume that it is D_e = 200 mm, D_i = 100 mm. With these values, the tangential stress, at the intrados (d = 100) and at the extrados (d = 200), is worth:

$$\sigma_{ti} = 83 \text{ MPa}$$
$$\sigma_{te} = 33 \text{ MPa}$$

These are the analytical values to be compared with the numerical calculation.

Figure 1.7 contains the plane strain finite element model for a section of the cylinder.