7. INTRODUCTION TO THE FINITE ELEMENT METHOD

Engineers use a wide range of tools and techniques to ensure that the designs they create are safe. However, accidents sometimes happen and when they do, companies need to know if a product failed because the design was inadequate or if there is another cause, such as an user error. But they have to ensure that the product works well under a wide range of conditions, and try to avoid to the maximum a failure produced by any cause. One important tool to achieve this is the finite element method.

"The finite element method is one of the most powerful numerical techniques ever devised for solving differential (and integral) equations of initial and boundary-value problems in geometrically complicated regions." (Reddy, 1988). There is some data that can not be ignored when analyzing an element by the finite element method. This input data is to define the domain, the boundary and initial conditions and also the physical properties. After knowing this data, if the analysis is done carefully, it will give satisfactory results. It can be said that the process to do this analysis is very methodical, and that it is why it is so popular, because that makes it easier to apply. "The finite element analysis of a problem is so systematic that it can be divided into a set of logical steps that can be implemented on a digital computer and can be utilized to solve a wide range of problems by merely changing the data input to the computer program." (Reddy, 1988).

The finite element analysis can be done for one, two and three-dimensional problems. But generally, the easier problems are those including one and two dimensions, and those can be solved without the aid of a computer, because even if they give a lot of equations, if they are handled with care, an exact result can be achieved. But if the analysis requires three-dimensional tools, then it would be a lot more complicated, because it will involve a lot of equations that are very difficult to solve without having an error. That is why engineers have developed softwares that can perform these analyses by computer, making everything easier. These softwares can make analysis of one, two and three dimensional problems with a very good accuracy.

A basic thing to understand how finite element works is to know that it divides the whole element into a finite number of small elements. “The *domain* of the problem is viewed as a collection of nonintersecting simple subdomains, called *finite elements*… The subdivision of a domain into elements is termed finite element discretization. The collection of the elements is called the finite element mesh of the domain.” (Reddy, 1988). The advantage of dividing a big element into small ones is that it allows that every small element has a simpler shape, which leads to a good approximation for the analysis. Another advantage is that at every node (the intersection of the boundaries) arises an interpolant polynomial, which allows an accurate result at a specific point. Before the finite element method, engineers and physicians used a method that involved the use of differential equations, which is known as the finite difference method.

The method of the finite element is a numerical technique that solves or at least approximates enough to a solution of a system of differential equations related with a physics or engineering problem. As explained before, this method requires a completely defined geometrical space, and then it would be subdivided into small portions, which together will form a mesh. The difference between the method of the finite element and the method of the finite difference is that in the second one, the mesh consists of lines and rows of orthogonal lines, while in the method of the finite element the division does not necessarily involves orthogonal lines, and this results in a more accurate analysis (Figure 38).

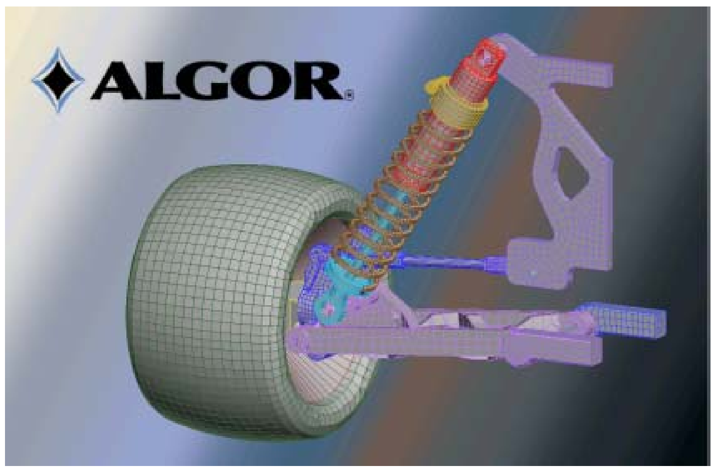


Figure 38 (taken from Algor 13 ®)

The equations used for the finite element method are a lot, but they have the basis on some single equations that describe a particular phenomenon. Those equations are:

The elliptic equation is described by $\frac{\partial^2 \Phi}{dx^2} + \frac{\partial^2 \Phi}{\partial y^2} = 0$

The parabolic equation is described by $\frac{\partial^2 \Phi}{\partial x^2} - \frac{\partial \Phi}{\partial t} = 0$

The hyperbolic equation is described by $\frac{\partial^2 \Phi}{\partial t^2} - \frac{\partial^2 \Phi}{\partial x^2} = 0$ [1]

No matter which is the cause of the internal forces and the deformation that they cause, there are three basic conditions that allow the finite element analysis: the equilibrium of forces, the compatibility of displacements and the laws of material behavior. “The first condition merely requires that the internal forces balance the external applied loads.” (Rockey et. al., 1983). That is the most important condition, but the other two assure that the system will be a statically determinate problem. Another condition that must be taken into account is that there exists a relationship between the load applied and the deformation, and this is given by Hooke’s law, as explained in past chapters, but only in the elastic range.

In order to achieve a structural analysis by matrix methods, there might be three ways: stiffness (displacement) method, flexibility (force) method and mixed method. In the first two methods, two basic conditions of nodal equilibrium and compatibility must be reached. In the first method, the once the displacement compatibility conditions are reached, then an answer can be given. In the second method, once the conditions of nodal equilibrium are satisfied and then the compatibility of nodal displacement, forces are known in the members.

One of the principles that is the basis of the finite element method is the one known as principle of virtual work. “This principle is concerned with the relationship

[1] Formulas taken from: Sabonnadière, Jean Claude, “Finite Element Methods in CAD: Electric and Magnetic Fields”, Springer-Verlag New York Inc, United States, English Edition, 1987, p. 1.

which exists between a set of external loads and the corresponding internal forces which together satisfy the equilibrium condition, and also with sets of joint (node) displacements and the corresponding member deformations which satisfy the conditions of compatibility.” This principle can be stated in terms of an equation of equilibrium of loads, where the work done by the external loads is equal to the internal virtual work absorbed by the element. Or expressed as an equation:

$$\Sigma F \cdot \delta = \int^{v} \sigma \cdot \varepsilon \cdot d$$

where F are the external loads,   the deflection,   the system of internal forces, and the internal deformations.

A pin ended tie has similar characteristics to those of an elastic spring that is subjected on one end, looking downwards, suffering the effects of gravity and the effect of an external load. The direct relationship between the force and the displacement of the free end is:

$$F = k \cdot \delta$$

the vale k is known as the stiffness of the spring. Once the value of the applied force and the value of the stiffness, the equation can be inverted to find the displacement:

$$\delta = \frac{1}{k} F$$

This is a simple example for systems that imply only a few data. But when the problem implies more complex systems, then the equations become a little more complicated. When a number of simple members are interconnected at a number of nodes, the displacement caused by the load can only be described by simultaneous equations. Then, the simpler equation seen before becomes:

$$\{F\} = [K]\{\delta\}$$

where K is the stiffness of the whole structure.

For example, for a spring that has two pins, it generates two forces and two displacements. Therefore, the stiffness matrix would be of order 2 x 2:

$$\begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix} = \begin{bmatrix} k_{11}k_{12} \\ k_{21}k_{22} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix}$$

where *u* represents the displacement.

Boundary conditions are the limitations set for the problem. These limitations are necessary in order to solve it, because otherwise, the system would be taken as a rigid body. The limitations stated by these boundary conditions are like where does the element is likely to move, and were it is restricted. If there were not boundary conditions, the body would be floating in the space, and under the action of any load, it would not suffer any deformation, but it would move around the space as a rigid body. So, when assuming boundary conditions, it has to be assured that the element has enough of them in order to prevent moving as a rigid body. Once this is done, the values of the displacements are obtained and can be substituted in the last equation seen, which will give that the displacement is equal to zero, because the element can not move in any direction. Then, algebra is applied, and the values of the forces or of the stiffness are known.

7.1 Applications of the finite element method in engineering

Every designed made has to fulfill certain specifications, and among them is working under a variety of conditions: temperature, humidity, vibrations, etc. The job of the designer is to achieve this, and to assure that the product will work effectively, tanking care of the user and of the element. An engineer has to follow certain steps in order to create a good product with high quality. First, the steps of the design flow chart must be followed:

1. recognize a need
2. specifications and requirements
3. feasibility sturdy
4. creative design and development
5. detailed drawings
6. prototype building and testing
7. design for manufacture

These are the basic concepts of the design, but it involves a lot of other things to be able to assure a good performance of the product. After the designer has the calculations of the dimensions, tolerances, manufacturing parameters, etc, some other tests should be done. For example, an engineer has to know whether the product is going to support certain loads, or how it is going to behave with temperature variations, or what could happen if vibrations are present. This is where the finite element method enters in action in engineering.

In the last pages, it has been explained how the finite element works and which are the basis of it. Now it is time to explain what is it good for, the applications, the benefits, etc. For example, the first application of the method was introduced by Richard Courant to solve torsion on a cylinder. Then, in the middle 50's, the method started to be applied for airframe and structural analysis, and then used in civil engineering. In general, finite element methods are used in a wide variety of engineering applications, like in computer graphics, heat transfer, electrical and magnetic fields, among others.

Use of finite element method in mechanical engineering is very wide. For example, it is used in mechanics of materials, for structures and trusses. It is used to understand and to prevent how some structures are going to behave under the action of some loads. For example, for a bridge, how is it going to behave with the vibrations, or with the effect of the air, or with the variations of temperature. The aircraft industry uses this method to determine the static and dynamic answer of planes and space crafts to the great variety of environments and conditions that can be found during their operation.

In the case of mechanics of fluids, the method is used to know how a wing of a plane is going to behave with the air flux passing through it, if it is going to resist, how much vibration is this going to cause, in order to avoid resonance (because of the vibrations caused by the turbines too). The finite element method allows calculating the drag and the flotation forces caused during operation.

For heat transfer, it allows to know how a turbine is going to behave, and how the material is going to be affected for the effect of the heat. It is known that heat also creates stresses, and this is a fundamental concept when talking about design of turbines. Another important thing to mention here is that when turbines are working, they reach very high temperatures, and they use a coolant to maintain the temperature under some point. This coolant, when touching the hot turbine, creates a thermal shock, which also produces stresses. These stresses can be measured through the finite element method. Something else that it is important is that some components of the turbine (like blades) have holes to let the air in to act as a coolant to avoid overheating. These holes can act as stress concentrators, and by this method, it can be analyzed which is the way they affect less the stress distribution.

One important area where this method is used is in design of mechanical elements. Generally, these elements have to support high loads, whether they are radial, tension, compression, tangential, etc. Sometimes these loads are combined, which makes the element designed more prone to failure. Most of the time, mechanical parts also have holes for assembly, for flow of a certain fluid, etc, and these holes create stress concentrations. A very specific and careful analysis has to be made. Here is where the finite element analysis plays an important role. For example, in the design of a crank, it is subjected to different loads, and it has specific boundary conditions. It is tested with total restriction on one side, and with a load applied tangentially on the other side. The results are like in Figure 39, where the areas that have bigger stresses can be identified according to the color specified on the right of the screen.

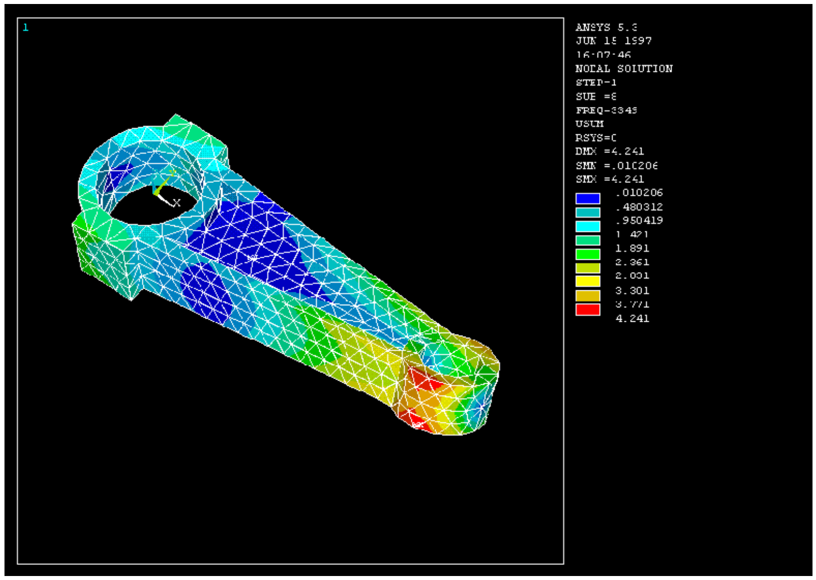


Figure 39

Other application in design is in the design of a control arm. This is for a combination of mechanics with electronics, but it still has forces applied, and holes manufactured for assembly. The boundary conditions have to be established according to the allowable movements of the robot, and then the loads applied in the points where it carries something. The results are those in Figure 40

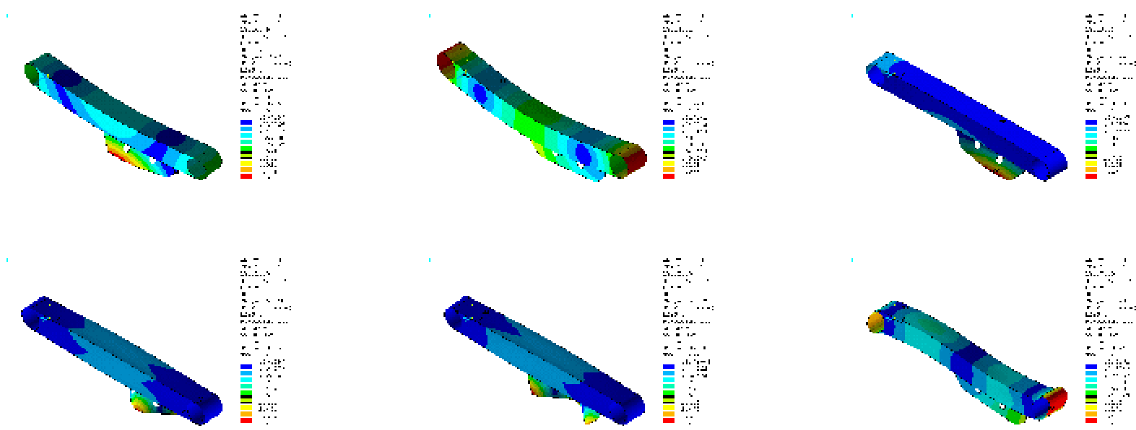


Figure 40

The automotive industry uses this method to see how the cars behave under some conditions of load, and for crash tests. For this industry, it has a lot of advantages. After the car is modeled in a CAD software, it is exported to a finite element analysis software, and after all the conditions are set, the test begins. The method has the advantage of giving very accurate information. For example, the deformation suffered point by point (node by node) at a specific time. The fact of they doing these type of analyses represents a great advantage for them, because they save money, they save time, and they also save material, and yet they have very reliable information of the behavior of the car. For example, Figure 41, which is a model of a thesis done the last semester, where the student was analyzing the effect of the impact on the car, and also on a dummy. He could calculate the stress at any point he wanted to, and at any time of the collision, and he was able to determine which areas of the car and the dummy were the most affected with the impact. Another example is Figure 42.

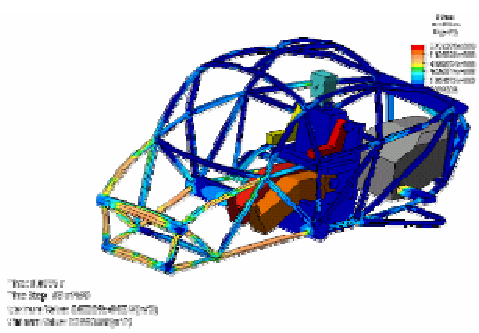
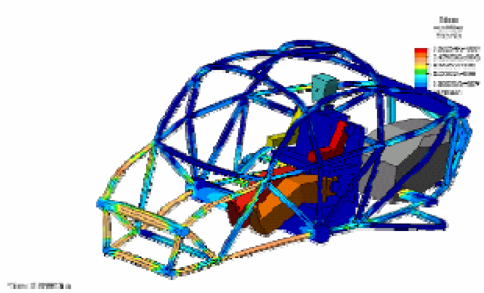


Ilustración 7-48 Colisión 14 – 02.



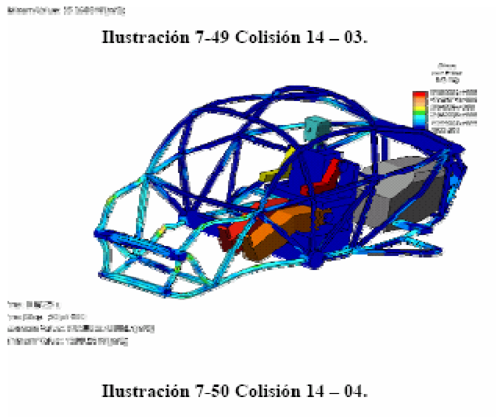


Figure 41 (from Alfredo Pérez Mitre)

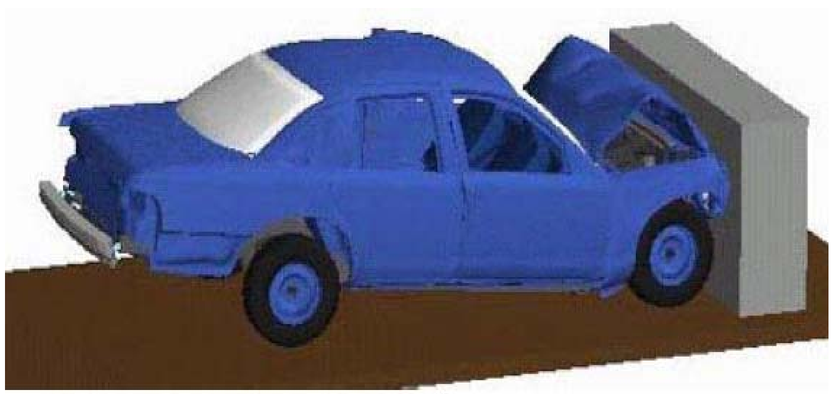


Figure 42

7.2 Advantages

The use of the finite element method has a lot of advantages, and most of them have already been commented in this thesis, but here it is going to be made a short condensation of them.

First, it is a very important tool for stress and strain analysis, not only because it provides accurate information, but because it also saves a lot of money and time by simulating the events in computer and not in real life. It is a really easy to use tool. Once some tutorials are followed, the only thing left to do is to explore a little bit of the software, and then to apply all the knowledge acquired. The principles of the method are easy to understand, even if when the model is complex, the analyses are also complex.

For every engineer it is a very reliable tool, because it is very specific for each occasion, and it is able to perform different analyses of the same model under different circumstances with simple changes in the boundary conditions, in the loads, in the material, or whatever the problem demands.

The method can help to modify each design in order to increase the service life of it as much as possible. That is the case of study of this thesis, where some analyses are run in the computer to see how the presence of a stress concentrator affects the behavior of the element. If the stress concentration is high, the element can be modified with ease and then subjected to analysis again, and depending on the results, a decision has to be taken to see if it needs more changes or if it has reached or if it is even close to its maximum service life condition.

The combination of this as a software with other types of softwares is a very useful tool, because the program of finite element analysis allows the designer to import models from other CAD softwares, and that way it does not forces the engineer to make the solid in the FEA CAD. That way, complex models can be created and analyzed with the combination of those two powerful tools.