2. GENERAL PROBLEM DEFINITION

2.1. General considerations on the method

The computer-aided design of a mechanical system involves identifying the shape and size of its constitutive elements by using advanced software in solid modeling, in the analysis of physical processes, in the synthesis and/or optimization.

The mechanical system of a vehicle, installation, machinery, robot, aircraft, etc. can be divided in assemblies which in turn are made of subassemblies and distinct constitutive parts, called machine parts. Subassemblies can also be made up of other subassemblies or other distinct constitutive parts. Both the assemblies and the subassemblies of the mechanic system are standalone entities, useful for the structural study of the system and for the technical optimization of the assemblage. The components of a mechanical system (or subsystem) are in permanent, direct fixed interaction (removable or non-removable) or direct moving interaction (without lubrication) or indirect (with lubrication). There are many types of such connections, in terms of design and depending on functional and technological necessities.

The practice of designing and building mechanincal systems is in permanent development, constantly updading any performance achievements regarding the means, methods, possibilities and technologies available. In terms of functionality, different mechanical systems present certain machinery elements and/ or subassemblies which have identical or quasiidentical functions. Gradually, well-known design algorithms, as well as technologies specialized in executing and assemblying have been developed for these elements or subassemblies, seldom called machine parts.

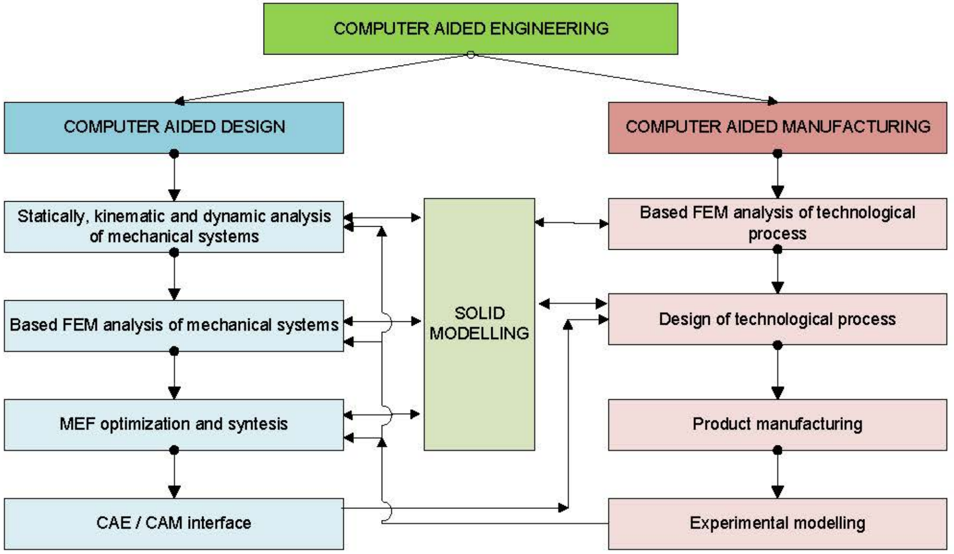


Fig. 11 The CAD-CAM module interface [40]

The emergence and booming development of numerical computer performance in the last decades allowed to obtain advanced software which implies new possibilities of modeling, analysis and synthesis of subassemblies, of elements and/ or machine parts. Most of these advanced programs rely on numerical methods, including the best knownn one, the Finite Element Method in the practice field of physical phenomena analysis.

Fig. 11 presents a general outline of the main activities for designing and implementing mechanical products, especially aiming at the identification of a hierarchy of the programs based on FEM.

Solid modeling is the sum of the activities describing objects in terms of geometry and physics or even of spatial domains in order to create a representation of them using the numerical computer. The geometric shapes of the systemic studied elements can be modeled with certain degrees of idealization based on the current design stage. In the primary stages of complex mechanical systems, the degree of idealization related to modeling elements is increased, with the main objective of analyzing and sinthetizing on a main (functional) level, unlike the final stages, when modeling is made with as little deviations as possible from the nominal shape and size, where the main objective relates to the aspects of designing in detail.

The structural, static, kinematic and dynamic analysis of mechanical systems implies an ongoing study of the correlations between parameters and the actual characteristics and those imposed to the mechanical system, considering models with an increased degree of idealization for the constitutive elements. In order to arrive at the required characteristics, appropriate changes can be made, followed by reanalyzing or by synthesis and/ or optimization models. Consequent to these operations result the main dimensional and physical parameters of the elements and the subassemblies pertaining to the mechanical system.

The finite element analysis of simple elements or subassemblies of a mechanical system, using the results obtained in the previous stage, implies geometric remodeling and a detailed specification of the shape and in the same manner, finite elements modeling with increased accuracy. Some advanced programs based on FEM have special optimization and shape synthesis modules. Such programs allow generating geometric shapes which respect conditions of equal resistance, minimum volume or minumum mass. Furthermore, finite elements are commonly used also for the dynamic study of mechanic systems with elastic deformable elements.

The CAD-CAM module interface (Fig. 11) connects the engineering design software and the changes in shape and size in order to adjust to the technological processes available. The study of such processes (deforming, casting, diffusion, etc.) using finite element analysis allows to determine the shape and dimensional parameters necessary for design devices and for the establishment of the optimal technological regimes.

It is possible to rapidly achieve mechanical material products with high performance by massive introduction of numerical computers with advanced programs for both dimensional synthesis and implementation. The stages of computer-aided design and manufacturing of technical systems modifies perpetually and is constantly being updated in accordance with the progress in the field of modeling, analysis, as well as the development of techical performance of computing systems.

In order to obtain professional products, modeling and theoretical analysis of real phenomena can be done through two main directions: by studying theories on general situations and practical studies through the analysis of concrete practical cases. Fig. 12 outlines the main steps listed in both directions. It also shows that the theoretical analyses aims at the practical results is performed based on fundamental studies. General theoretical analysis of real phenomena are based on computational theoretical models that are assigned appropriate mathematical models.

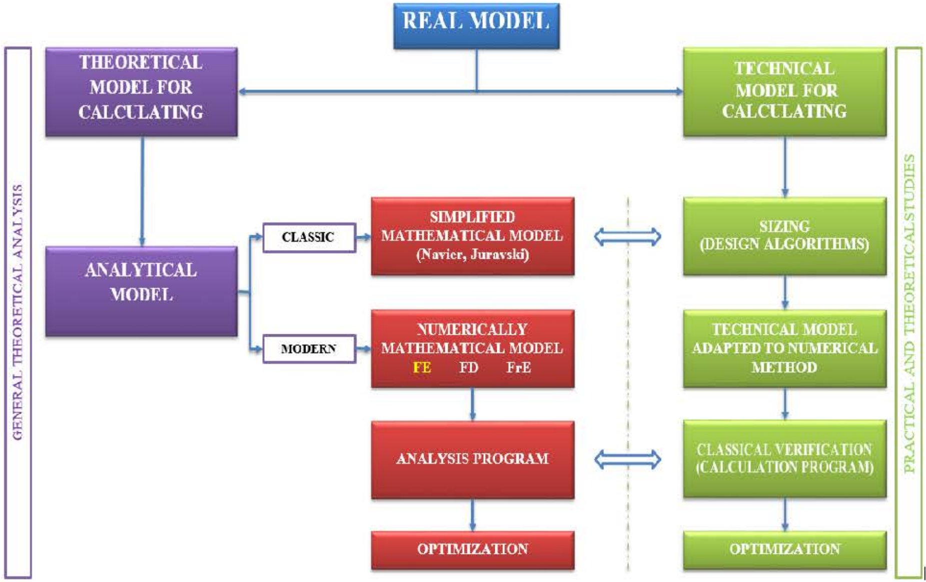


Fig. 12 Theoretical and practical approaches [40]

The computational theoretical model (Fig. 12) is an approximation of the real model which involves identifying the shape and size of the geometric domain and the physical parameters known as the qualitative indication of unknown physical parameters. For known geometrical and physical parameters, the variational functions and their limit values are being established.

The analytical mathematical model (Fig. 12), associated with the computational model made, is in most cases a system of differential and/ or integro-differential equations, with complementary sets of appropriate initial and boundary conditions. In many cases it is possible to describe mathematically physical phenomena, synthetized in the computational model using a variational calculus through a functional. This description, often used due to the simplicity of the methods and algorithms for solving the mathematical model, developed in various forms, especially for mechanical engineering problems.

To solve by the classical approach, following rough approximations regarding geometry, initial and boundary conditions, and material properties applied on theoretical models of mathematical calculus, we obtain simplified analytical mathematical models which can be processed using the manual calculus, slide rule or the calculator.

For example, the calculus model of bendable mechanical structures, with the methods of the theory of elasticity and strength of materials, we obtain specific mathematical models leading to simple calculus relationships (Navier, Juravski, etc.) for different geometrical fields (bars, plates, shells, tubes, discs, etc..) and specific physical conditions.

In order to increase the precision of the results obtained by classical methods (Fig. 13), numerical methods through small, usually controllable approximations in respect to geometry, boundary conditions and material properties, lead to numerical modeling that can be solved only by numerical computer. The practice of numerical modeling which involves the study of physical phenomena in continuous environments by splitting them into smaller subdomains called finite elements, developed and became a business performance programs (NASTRAN, ANSYS, ALGOR, COSMOS, CATIA, etc.) that have pre and post-processors with advanced facilities of data input and processing.

Theoretical and practical studies applicable in the design of specific machine elements are based on the techincal calculus model. Since advanced programs that are based on FEM deal with analysis, preliminarily, sizing calculation is required (predimensioning) using, in particular, traditional methods of strength of materials. In order to use advanced software to analyze and optimize the shape of the machine elemet structure, predesigned in both shape and size, it is necessary to complete one or more models of analysis adjusted to the numerical method on which the program is based.

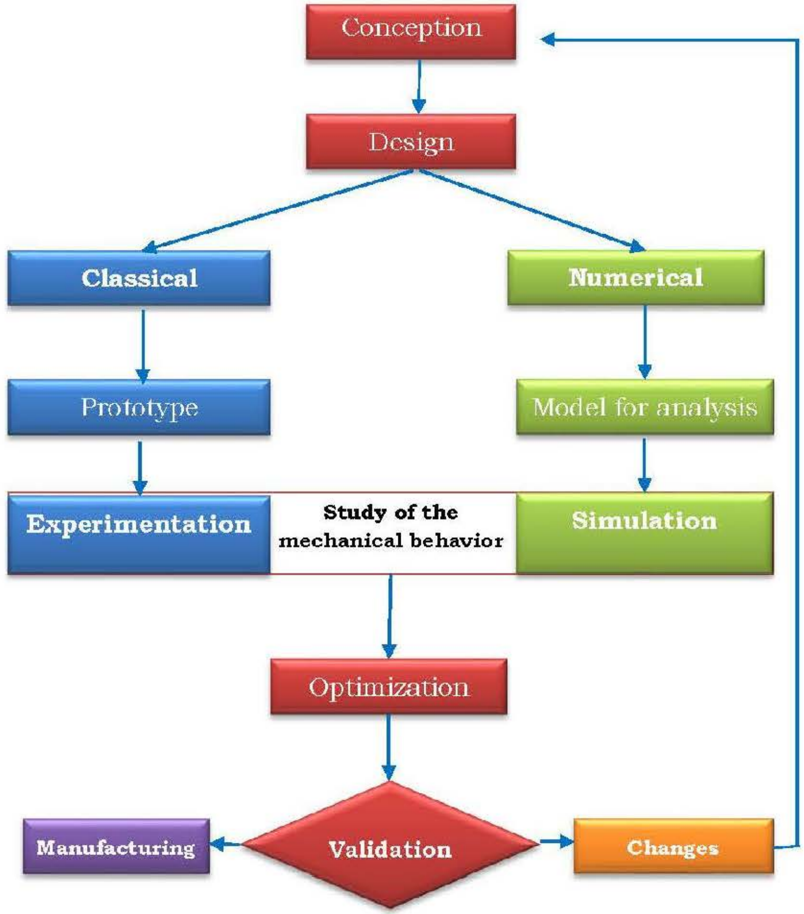


Fig. 13 Classical and numerical analysis

This paper particularly seeks to explore the finite element analysis software for the advanced design of machine elements (organs) and/ or mechanical system subassemblies. Various finite element analyses presented are based on the definition of the problem in the general context of the design and ends with the visualization, analysis and synthesis of the results.

These analyses are performed in a CATIA environment, characterized by a high level of integration of CAD and CAE modules, as it was shortly highlighted in the first chapter. So, as of late, thanks to this integration and high level of communication of the human operator with the programming environment, the design engineers can easily access CAE modules (Computer Aided Engineering) of finite element analysis. This requires that designers have knowledge of dealing with finite element analysis models and processing results. This chapter presents the general problems related to finite element modeling including the geometry, the material properties, the boundary conditions and commonly encountered difficulties in various situations in practice.

2.2. FEM terminology[1] [17]

The "degrees of freedom" term, as well as "stiffness matrix" and "force vector," originated in structural mechanics, the application for which FEM was invented. These names have carried over to non-structural applications. Classical analytical mechanics is that invented by Euler and Lagrange in the XVIII century and further developed by Hamilton, Jacobi and Poincar′e as a systematic formulation of Newtonian mechanics. Its objects of attention are models of mechanical systems ranging from material particles composed of sufficiently large number of molecules, through airplanes, to the Solar System. The spatial configuration of any such system is described by its degrees of freedom or DOF. These are also called generalized coordinates. The terms state variables and primary variables are also used, particularly in mathematically oriented treatments [17].

If the number of degrees of freedom is finite, the model is called discrete, and continuous otherwise. Because FEM is a discretization method, the number of DOF of a FEM model is necessarily finite. They are collected in a column vector called $\boldsymbol{u}$. This vector is called the DOF vector or state vector. The term nodal displacement vector for u is reserved to mechanical applications. In analytical mechanics, each degree of freedom has a corresponding "conjugate" or "dual" term, which represents a generalized force. In variational mathematics this is called a

[1] *The content of this chapter (marked with [17]) was taken from the paper Felippa, C.A.: Introduction to Finite Element Methods, lecture notes, with the written consent of the author, whom I thank.*

duality pairing. In non-mechanical applications, there is a similar set of conjugate quantities, which for want of a better term are also called forces or forcing terms. They are the agents of change.

These forces are collected in a column vector called f. The inner product $f^T u$ has the meaning of external energy or work. Energy is the capacity to do work. Thus energy and work potentials are the same function (or functional), but with signs reversed. Just as in the truss problem, the relation between u and f is assumed to be of linear and homogeneous. The last assumption means that if u vanishes so does f. The relation is then expressed by the master stiffness equations:

$$Ku = f. \qquad \text{Eq. 1}$$

K is universally called the *stiffness matrix* even in non-structural applications because no consensus has emerged on different names [17].

The physical significance of the vectors u and f varies according to the application being modeled, as illustrated in Table 1. If the relation between forces and displacements is linear but not homogeneous, equation (Eq.1) generalizes to

$$Ku = f_M + f_I . \qquad \text{Eq. 2}$$

Here f_I is the initial node force vector and f_M is the vector of mechanical forces.

Table 1 Significance of u and f in Miscellaneous FEM Applications [17]

Application Problem	State (DOF) vector u represents	Conjugate vector f represents
Structures and solid mechanics	Displacement	Mechanical force

Heat conduction	Temperature	Heat flux
Acoustic fluid	Displacement potential	Particle velocity
Potential flows	Pressure	Particle velocity
General flows	Velocity	Fluxes
Electrostatics	Electric potential	Charge density
Magnetostatics	Magnetic potential	Magnetic intensity

2.3. Idealization [17]

Idealization passes from the physical system to a mathematical model. This is the most important step in engineering practice, because it cannot be "canned." It must be done by a human.

2.3.1. Models

The word "model" has the traditional meaning of a scaled copy or representation of an object. And that is precisely how most dictionaries define it. We use here the term in a more modern sense, which has become increasingly common since the advent of computers:

A model is a symbolic device built to simulate and predict aspects of behavior of a system.

Note the distinction made between behavior and aspects of behavior. To predict everything, in all physical scales, you must deal with the actual system. A model abstracts aspects of interest to the modeler. The qualifier symbolic means that a model represents a system in terms of the symbols and language of another discipline. For example, engineering systems may be (and are) modeled with the symbols of mathematics and/or computer sciences.

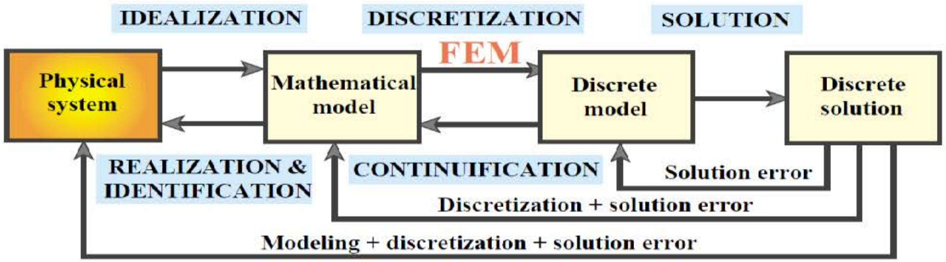


Fig. 14 A simplified view of the physical simulation process [17]

2.3.2. Mathematical Models

Mathematical modeling, or idealization (Fig. 14), is a process by which an engineer or scientist passes from the actual physical system under study, to a *mathematical model* of the system, where the term model is understood in the sense of previous definition.

The process is called *idealization* because the mathematical model is necessarily an abstraction of the physical reality. The analytical or numerical results produced by the mathematical model are physically re-interpreted only for those aspects.

To give an example of the choices that an engineer may face, suppose that the structure is a flat plate structure subjected to transverse loading. Here is a non-exhaustive list of four possible mathematical models:

1. A *very thin* plate model based on Von Karman's coupled membrane-bending theory.
2. A *thin* plate model, such as the classical Kirchhoff's plate theory.
3. A *moderately thick* plate model, for example that of Mindlin-Reissner plate theory.
4. A *very thick* plate model based on three-dimensional elasticity.

The person responsible for this kind of decision is supposed to be familiar with the advantages, disadvantages, and range of applicability of each model. Furthermore the decision may be different in static analysis than in dynamics.

Why is the mathematical model an abstraction of reality? Engineering systems, particularly in Aerospace and Mechanical, tend to be highly complex. For simulation it is necessary to reduce that complexity to manageable proportions. Mathematical modeling is an abstraction tool by which complexity can be tamed.

Complexity control is achieved by "filtering out" physical details that are not relevant to the design and analysis process. For example, a continuum material model filters out the aggregate, crystal, molecular and atomic levels of matter. Engineers are typically interested in a few integrated quantities, such as the maximum deflection of a bridge or the fundamental periods of an airplane.

Although to a physicist this is the result of the interaction of billions and billions of molecules, such details are weeded out by the modeling process. Consequently, picking a mathematical model is equivalent to choosing an information filter.

2.3.3. Implicit vs. Explicit Modeling

As noted the diagram of (Fig. 14) is an oversimplification of engineering practice. The finite element method (FEM), or finite element analysis (FEA), is based on the idea of building a complicated object with simple blocks, or, dividing a complicated object into small and manageable pieces. Application of this simple idea can be found everywhere in everyday life as well as in engineering.

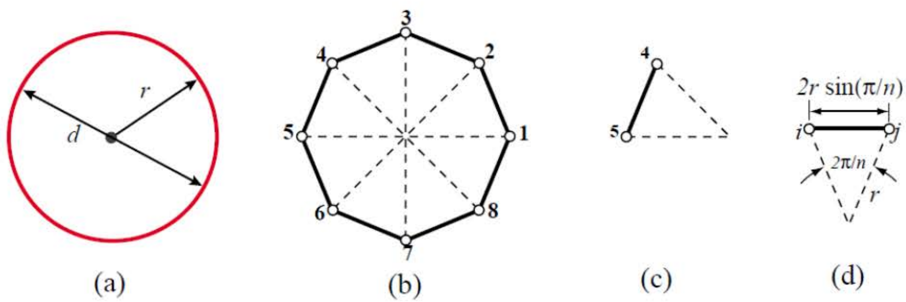


Fig. 15 The "find π" problem treated with FEM concepts: (a) continuum object, (b) a discrete approximation by inscribed regular polygons, (c) disconnected element, (d) generic element

Examples:

- Lego (kids' play)
- Buildings
- Approximation of the area of a circle (Fig. 15).

The more common scenario is that pictured in (Fig. 16) and (Fig. 17).

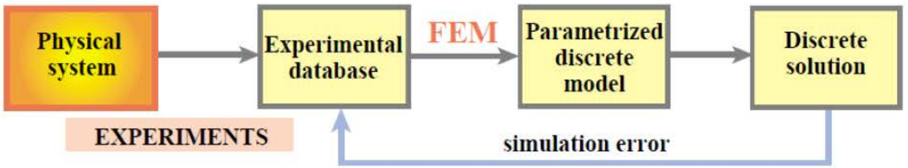


Fig. 16 The physical simulation process [17]

A common scenario in industry is: you have to analyze a structure or portion(s) of one, and at your disposal is a "black box" general-purpose finite element program. Those programs offer a catalog of element types; for example, bars, beams, plates, shells, axisymmetric solids, general 3D solids, and so on. The moment you choose specific elements from the catalog you automatically accept the mathematical models on which the elements are based. This is implicit modeling. Ideally you should be fully aware of the implications of your choice. Providing such "finite element literacy" is one of the objective of this book. Unfortunately many users of commercial programs are unaware of the implied-consent aspect of implicit modeling and their legal implications.

The other extreme happens when you select a mathematical model of the physical problem with your eyes wide open and then either shop around for a finite element program that implements that model, or write the program yourself. This is explicit modeling. It requires far more technical expertise, resources, experience and maturity than implicit modeling. But for problems that fall out of the ordinary it could be the right thing to do.

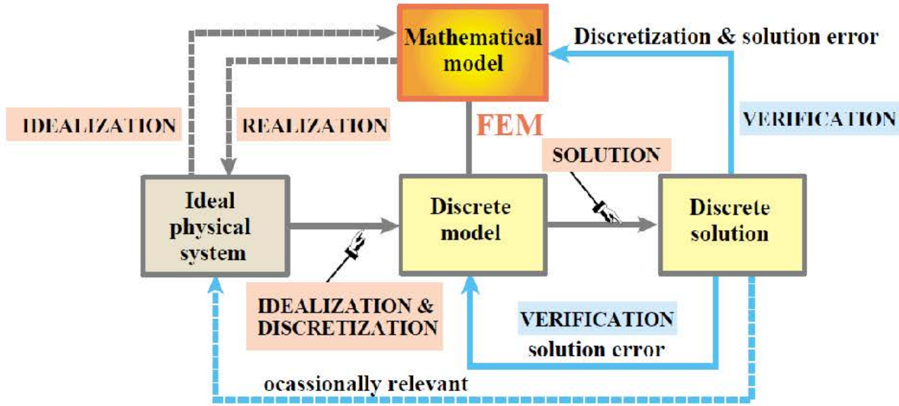


Fig. 17 The Physical FEM. The physical system (left box) is the source of the simulation process. The ideal mathematical model (should one go to the trouble of constructing it) is inessential [17]

In practice a combination of implicit and explicit modeling is common. The physical problem to be simulated is broken down into subproblems. Those subproblems that are conventional and fit available programs may be treated with implicit modeling, whereas those that require special handling may only submit to explicit modeling.

2.4. The theoretical quasi-general model for finite element analysis of a mechanical subassembly element

Fig. 18 shows a theoretical quasi-general model of finite element analysis of a subset of a mechanical element [40]. The continuous structure of this element, finite volume V and surface **S**, is made of solid materials with different behavior (linear, nonlinear), described by a specific laws. For the material or materials used, the values of density, the mechanical properties (density, elasticity matrices, damping factor etc.) and thermal (thermal expansion coefficient, specific heat, etc.) and the allowable resistances (traction, compression, usually and shearing) are known.

Over the structure of the analyzing element act the following external forces: generalized forces (**P_i** forces and / or moments, **$i = 1, 2 ... m$**) concentrated in points, generalized forces distributed on a line (forces or moments **q**, on the **C_p** line), on an area (the forces and / or p moments on the **S_p** surface) and in volume (mass forces **f_g**, centrifugal **t** and / or the inertial **f_j**).

The structure of the analyzing element operates in a limited temperature range between T_0 initial temperature and **T_f** final temperature. In addition, the structure may be under the action of thermal fields of temperature (on line distribution, on the surface or in volume) and/ or external thermal fluxes.

In the category of external loads are also included the elastic deformations required from certain areas of the structure through known values of δ_i displacements. This leads to an imposed shape of the deformed state of the structure area, which in Fig. 18, is synthesized by line **C_i**.

Direct interactions between the analyzed element structure with the structures of other elements of the mechanical subassembly can also be simulated using finite element analysis. These interactions may be permanent (void displacements usually imposed by boundary conditions) and / or temporary, also taking into account the friction (measured by the values of friction coefficients, μ) of the materials in the interacting areas and initial distances, **δ_0**.

The possibility of finite element analysis of the quasi-general model shown is conditioned by the existence of a set of imposed boundary conditions, usually synthesized by canceled shifts corresponding to certain points of the geometric field problem (e.g. surface area of Fig. 18). The solvability of finite element analysis model with loads and boundary conditions imposed is provided by the lack of possibilities in the structure’s kinematic movement.

Under the action of loads and imposed boundary conditions, the analyzed structure is deformed and within it, there are distributed internal forces called stresses. From a geometric perspective, displacement fields, the strains and stresses, are quantitatively described using the following displacement vectors:

$$[d] = [u \;\; v \;\; w]^T, \qquad \text{Eq. 3}$$

of the strains (Fig. 18, b),

$$[\varepsilon] = [\varepsilon x \ \ \varepsilon y \ \ \varepsilon z \ \ \gamma xy \ \ \gamma yz \ \ \gamma zx]^T \qquad \text{Eq. 4}$$

and namely, of the stresses (Fig. 18,c),

$$[\sigma] = [\sigma x \ \ \sigma y \ \ \sigma z \ \ \sigma xy \ \ \sigma yz \ \ \sigma zx]^T, \qquad \text{Eq. 5}$$

with the components connected to the tri-orthogonal straight coordinate system XYZ.

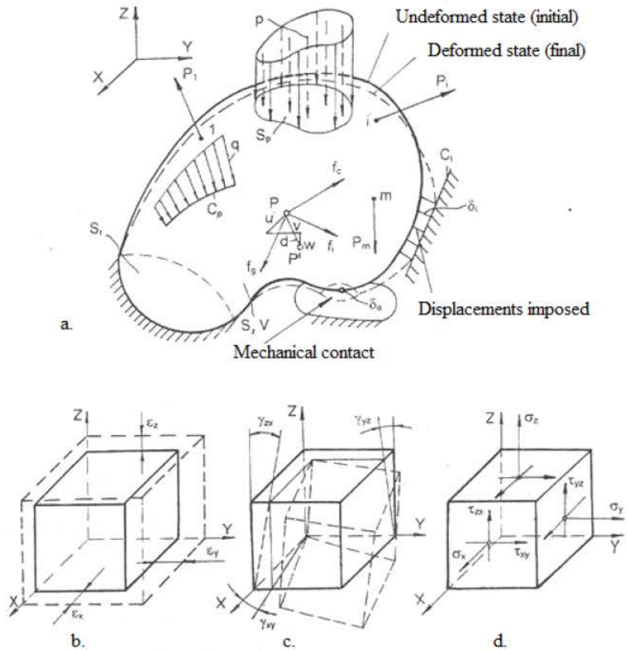


Fig. 18 Theoretical quasi-general model of finite element analysis [40]

Study using advanced programs based on FEM quasi-general model presented above and with given values for all input parameters, to constructively design the element to which is associated, usually aims to determine the fields of displacement, strain, stress, thermal and in addition, of parameters (force, displacement, temperature, etc.) from the connecting areas with other elements. The values and variations of these parameters, the functioning conditions and tolerated material characteristics make possible the evaluation of strength, rigidity and thermal characteristics of the structure analysis.

In the analysis and synthesis of mechanical assemblies elements it is unlikely to encounter an problem to which it is associated model described above. Usually, geometric shapes, loads and boundary conditions imposed to practical elements of mechanical subassemblies, are reduced to particular cases, leading to a simplified modeling, increased accuracy and a faster computational process. Based on these considerations, in the case advanced programs which are based on FEM, specific finite elements have been developed and software modules that allow solving the problems making connections with different possible particular cases.

2.5. Types of solvable problems using finite elements analysis

M.E.F. - Approximate solving method using a computer for a wide range of engineering problems:

- Equilibrum problems – determining unknown, time-independent parameters, for a steady state (linear or nonlinear static analysis, heat transfer analysis, the fluid flow or the magnetic field distribution).
- Custom values problems - determining certain critical values of the physical parameters, time invariables, equilibrium configurations and given boundary conditions (analysis natural frequency analysis, flexure, laminar flow regimes, resonance...).
- Propagation problems - unknown time-dependent parameters - the study of transient regimes (dynamic analysis of elastic and inelastic structures, heat transfer, unsteady flow).

Formulating an engineering problems involves:

- Identifying the type of problem;
- Identifying the working hypotheses adopted (the geometry in the problem's field, the material properties, variation field of main sizes, the functioning mode);
- Identifying the initial and boundary conditions.

In **Table 2** are presented the main types of solvable problems with advanced programs that are based on FEM, depending various criteria of quasi-general model customization. In practice, these problems can be encountered separately or in combination, following several criteria simultaneously customization.

Many practical applications with materializing in mechanical parts also include heat transfer processes and so in order to design is necessary to know the specific fields through thermal analysis. A part of the results of these tests, along with other types of loads can be considered for the analysis of mechanical components and/ or, sometimes, system subassemblies.

Table 2 Types of FEA problems [40]

Quasi-general problem	Customization criterion	Type of problem (analysis)
Advanced software analysis based on FEM of elements and mechanical systems	Type of fields	Thermal
		Mechanic
	Type of domain	Unidimensional
		Bidimensional
		Tridimensional
		Combined
	Dependancy on time variables	Static
		Dynamic
	Type of obtained values	Current
		Custom vectors
		Stability
	Dependancy between parameters	Liniar
		Nonliniar
	Possibility of considering connections	No connections
		With connections

The components of the mechanical systems have various three-dimensional (3D) shapes. In many practical cases, the shapes of elements are or may be considere two-dimensional (2D) - with one dimension much smaller than the other two - or one dimensional (1D) - where one dimension is much larger than the other two . For the finite element analysis of mechanical system assemblies with advanced programs based on FEM, without the detailed consideration of direct interactions between parts, often components can be of different forms (one-dimensional , two-dimensional and / or three-dimensional) . Starting from the possible forms of the domain of the studied element, finite element analysis can be three-dimensional, two-dimensional, one-dimensional or combined.

In terms of time-dependent loads, solving finite element model associated with the mechanical system element is called static analysis - without considering time as a variable - or dynamic analysis – with time-dependent unknown variables.

The study of the elements structure of mechanical systems with advanced software based on FEM which leads to the determination of the field displacements, strains, stresses and thermal as a result of the loads and the imposed boundary and the normal limit conditions, is considered normal analysis. In addition, using the same types of programs, in case of loading problems and abnormal boundary and limit conditions, limit states which may occur during operation can also be analyzed. In this sense, it is very common in the practice of design the stability analysis and also the analysis of vectors and their forms that lead to the underlying causes of critical load flexure and, respectively, the custom frequencies and the corresponding geometric configurations.

In terms of load-displacement dependencies and stress-strain can highlight the following types of analysis : linear, geometrically nonlinear, physically nonlinear (material) or geometrico-physically nonlinear. The first type of analysis is appropriate in cases of structures with small displacements when loads remain invariable during deformation and movement direction and can summarized, following a proportionality factor.

The second type of analysis corresponds to mechanical structures with large displacements, disproportionate with the loads, with the variable directions during deformation . The third type of analysis, unlike the first two, deals with the nonlinear, elastic and plastic behavior of a material, by means of suitable stress-strain characteristics. The last analysis is the general possible case when the two dependencies load-displacement and stress-strain are nonlinear.

Most finite element analyses of several element (subassembly) structures usually do not take into account the interactions between them, through the modeling of specific connecting phenomena, considered "frozen" by the continuity of the whole finite element structure at a nodal level. Starting from the importance of design processes (displacements, strains, stresses and frictions) from the connecting local areas, in the last years, there have been defined and implemented in many programs (including CATIA) specific connecting elements (translation coupling, rotation, roto-translation, rigid or elastic) that take into account the relative movements and contact elasticities that allow analysis links.

2.6. The model for analysis [40]

In order to efficiently and accurately simulate the behavior of mechanical systems or of subassembly elements with advanced programs that are based on FEM, a specific analysis model must be made. Finite element modeling, in many cases for analysis with the scope of designing, as a consequence of some features related to the geometric configuration, of the material behavior and physical phenomena, does not involve the consideration of the whole structure.

Minimizing the analysis models without sacrificing the accuracy of monitored parameters can be achieved by customizing the geometric configuration of inferior shapes (a 3D structure to a 2D or an 1D one, a 2D structure to 1D), by considering for modeling the area from the domain of the problem where the variations of unknown physical parameters are significant, and/ or by using symmetry.

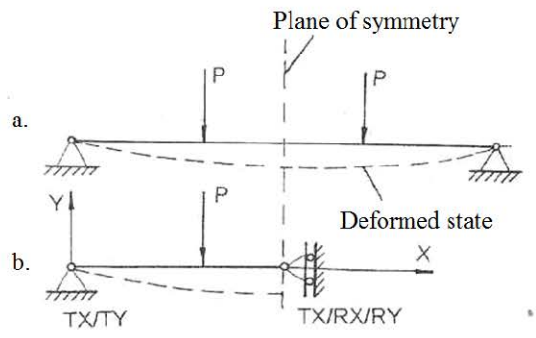


Fig. 19 Complete symmetry [40]

By customizing the geometric configuration of the structure to be analyzed, the conformity degree with the model reality decreases differently, in relation with both the values and variations of known and/ or unknown parameters. For example, for the 3D structure of the cylindrical piece, can be analyzed by reducing it to a 1D model, the accuracy of loading parameters at input and unknown parameters of the support and concentration area (dimensional jumps and keyways) decreases.

Through this customization, the finite element model size (the number of nodes) decreases considerably and considering the simplicity of the model in conjunction with the results obtained, it is seldom considered to be effective for checking the shafts of the standard broadcast. In the case of special transmissions for increased accuracy checks, a complex model (3D) analysis of the entire structure of the tree is made.

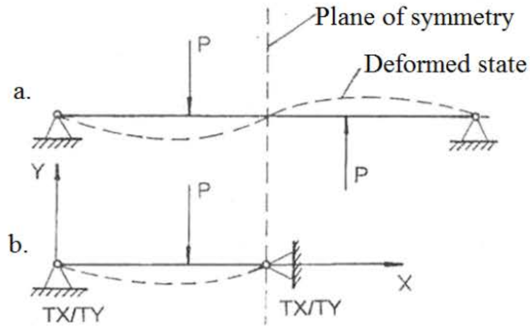


Fig. 20 Antisymmetry [40]

The structures of elements of mechanical systems to be analyzed with finite elements may have geometric, material, loading and/ or boundary conditions to a plane, two orthogonal planes, three orthogonal planes or even multiple plan symmetry.

For the purpose of creating a mechanical structure model analysis, taking into account the symmetry, it is necessary for the model to have a common symmetry regarding the geometry, the material properties and the imposed boundary conditions. In terms of loading, it is possible to create models as a result of complete symmetry (Fig. 19, b) when the loading has the same symmetry of geometry and boundary conditions (Fig. 19, a), or models (Fig. 20, b) generated by geometric and boundary conditions symmetry, and loading antisymmetry (Fig. 20, a).

Setting boundary conditions which take into account in the case of modelings which consider symmetries is being made by monitoring the accurate simulation of deformation processes from the initial structure. For example, the bar-like structure of Fig. 19,a, a full symmetry with a YZ parallel plane, the displacement after X is being canceled (TX) and rotations after the X and Y axes (RX, RY) or for the same structure (Fig. 20, a) but with an anti-symmetric load, the X and Y translations (TX, TY) are canceled.

The analysis of an asymmetric loading model shown in Fig. 21, a, in the case of the geometric symmetry structure and linear behavior, can be done by solving the model analysis associated to the half of the geometric domain for two sets of loads and boundary conditions corresponding to complete symmetry (Fig. 21, b) and the load antisymmetry (Fig. 21, c). The final state corresponding to the initial structure is obtained by summing the results for the two sets.

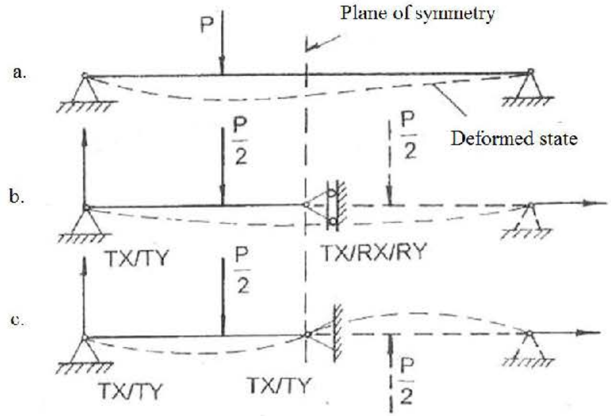


Fig. 21 Asymmetric loading [40]

Table 3. Mobilities of the joints [40]

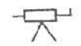
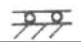
Canceled displacements	Displacement directions	Symbol	No. of canceled mobilities	No. of free mobilities
Rotation	Straight line or circular line		3	3
			2	4
			2	4
			1	5
Translation	Straight line		3	3
			2	4
			2	4
			1	5
	Circular line		3	3
			2	4
			2	4
			1	5

Table 4. Reactions and displacements in joints [40]

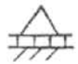
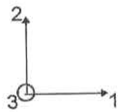
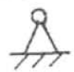
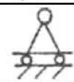
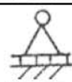
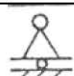
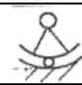
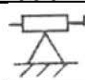
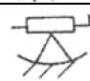
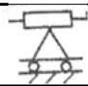
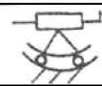
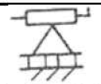
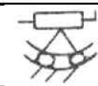
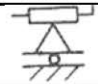
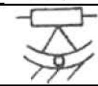
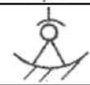
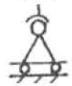
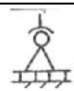
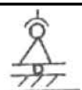
Coordinate system	Symbol: Straight axes	Symbol: Circular axes	Introduced reactions	Free Displacements
			F1/F2/F3/M1/M2/M3	-
			F2/F3/M1/M2/M3	T1
			F1/F2/M1/M2/M3	T3
			F2/M1/M2/M3	T1/T3
			F1/F2/F3/M1/M2	R3
			F2/F3/M1/M2	T1/R3
			F1/F2/M1/M2	T3/R3
			F2/M1/M2	T1/T3/R3
			F1/F2/F3/M1/M3	R2
			F2/F3/M1/M3	T1/R2

Table 4 (cont.)

			F1/F2/M1/M3	T3/R2
			F2/M1/M3	T1/T3/R2
			F1/F2/F3/M2	R1/R3
			F2/F3/M2	T1/R1/R3
			F1/F2/M2	T3/R1/R3
			F2/M2	T1/T3/R1/R3

In the case of symmetry of the structure to be analyzed using a plane, two planes or three planes (Fig. 22, a), the analysis model is reduced to half, a quarter or, respectively, an eighth (Fig. 22, b) of the geometric domain.

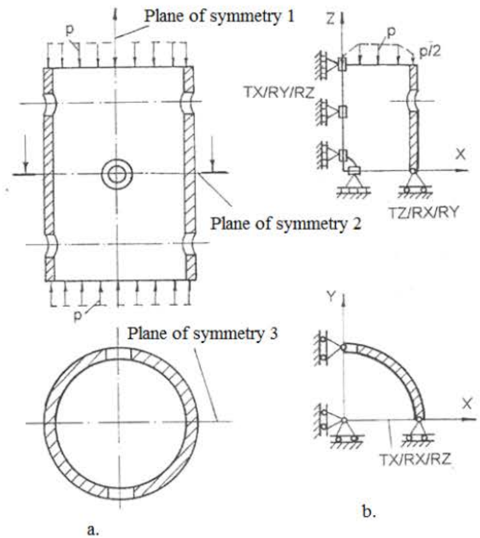


Fig. 22 Different types of symmetry [40]

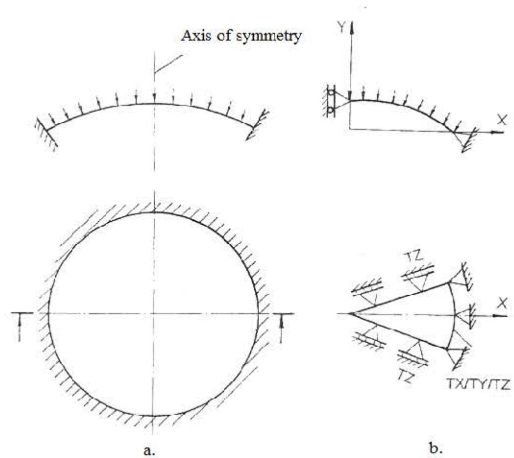


Fig. 23 Full simmetry to an axis [40]

Full symmetry to an axis, in a random case nonreductive to the axial-symmetric one (Fig. 23, a), involves the shaping of an angular sector (Fig. 23, b) or when the problem is of an axial-symmetrical nature, it leads to a plane model determined by the axial semisection by structure.

The problem of creating the optimal finite element analysis model is complex depending on the type of physical phenomena, the aimed requirements and performance of the program used

2.7. Samples of analysis models

The following are a few models for analysis: one-dimensional model – bars structure (Fig. 24), two-dimensional model – surface (Fig. 25), three-dimensional model – volumes (Fig. 26), model for thermic analysis (Fig. 27).

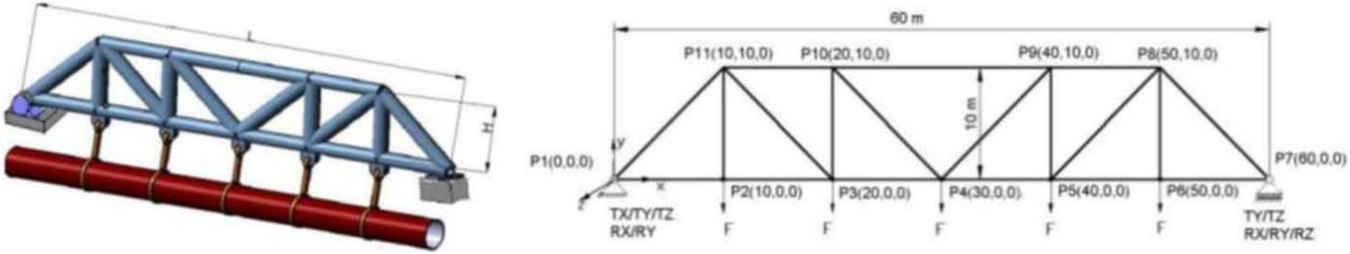


Fig. 24 Bars structure model for analysis (support beam)

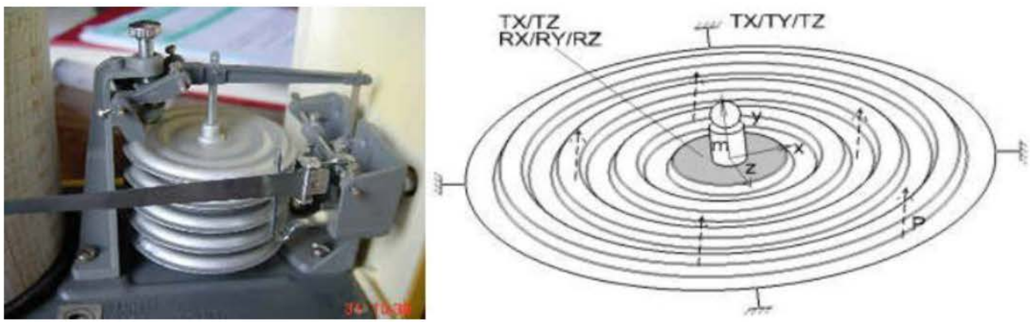


Fig. 25 Surface model for analysis (membrane)

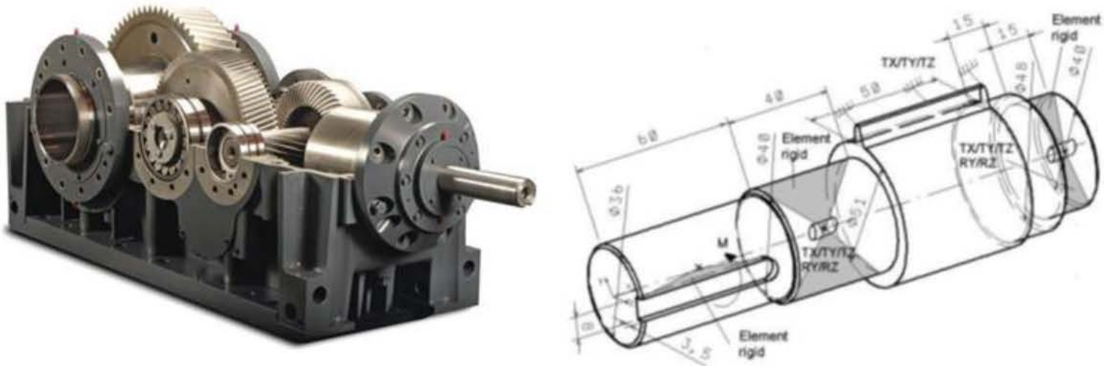


Fig. 26 Volume model for analysis (reducer shaft)

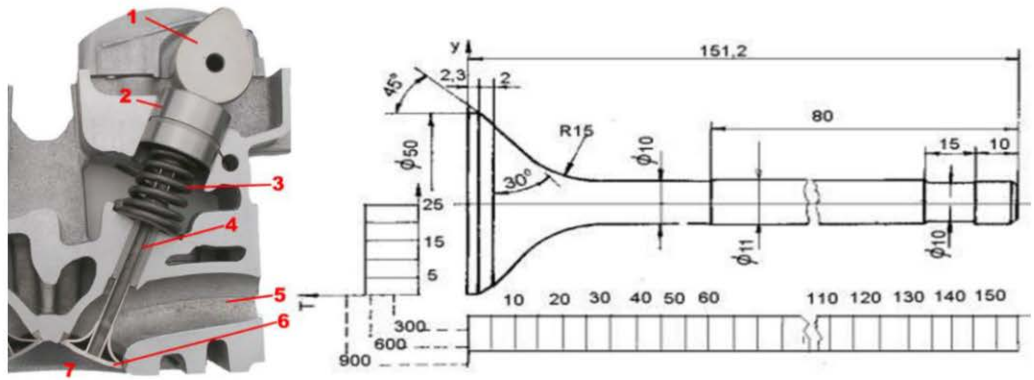


Fig. 27 Model for thermic analysis (engine valve)

2.8. The general procedure of FEA

In general, there are three phases (Fig. 28) in any computer-aided engineering task:

- Pre-processing – defining the model and environmental factors to be applied to it (typically a finite element model, but facet, voxel and thin sheet methods are also used).
- Analysis solver (usually performed on high powered computers).
- Post-processing of results (using visualization tools).

This cycle is iterated, often many times, either manually or with the use of commercial optimization software.

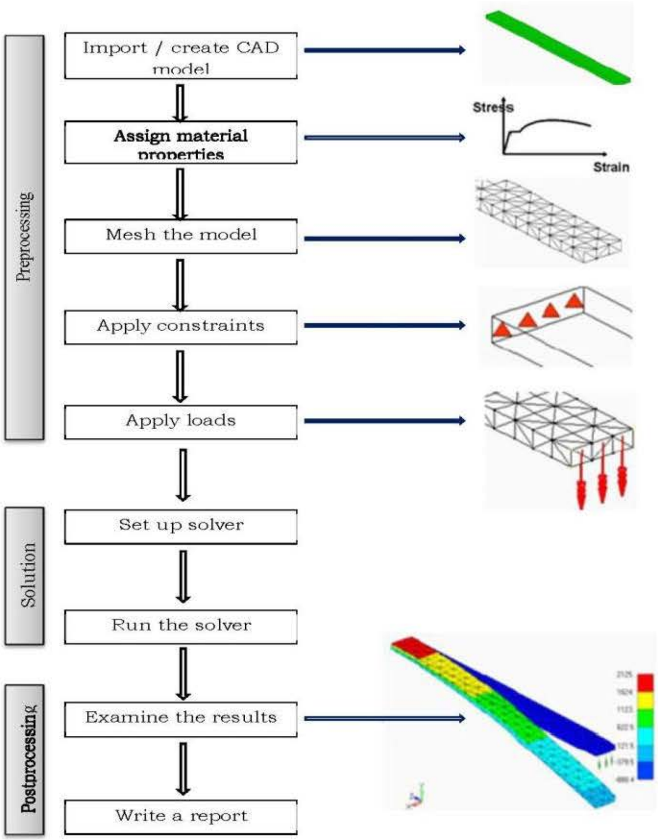


Fig. 28 Analysis procedure based on FEM [30]