Finite element analysis (FEA)

Finite element analysis (FEA) is the use of calculations, models and simulations to predict and understand how an object might behave under various physical conditions. Engineers use FEA to find vulnerabilities in their design prototypes. FEA uses the finite element method (FEM), a numerical technique that cuts the structure of an object into several pieces, or elements, and then reconnects the elements at points called nodes. The FEM creates a set of algebraic equations which engineers, developers and other designers can use to perform finite element analysis.

Frequently, the physical experiences of a product such as its structural or fluid behavior and thermal transport are described using partial differential equations (PDEs). Finite element analysis emerged as a way for computers to solve both linear and nonlinear PDEs. However, it is important to note that FEA only provides an approximate solution; it is a numerical approach to finding the real results of partial differential equations.

Using finite element analysis can reduce the number of physical prototypes created and experiments performed while also optimizing all components during the design phase. Finite element analysis software emerged in the 1970s with programs such as Abaqus, Adina and Ansys. Now, it is common to find virtual testing and design optimization integrated into the product development cycle to improve the product quality and reduce the time it takes to enter the market.

Types of FEA tests

Various different types of tests are used during finite element analysis, such as:

- **Structural static analysis**. This type of FEA analyzes a scaled model based on proportions. The test maintains that any structure that is sound on a small scale will be able to handle the same interactions with the full-scale structure and produce the same results.

- **Thermal engineering analysis.** This test explores variations in temperature and how it affects the design structure.

- **Modal analysis.** Every object vibrates at a frequency, so it is important to use modal analysis to test how disruptive external vibrations affect the product's structure. This form of finite element analysis also allows users to adjust for vibrations throughout the design stage, thus creating a strong final product.

- **Engineering seismic calculations.** This test helps developers understand the product's performance when dealing with various ground frequencies and vibrations, ensuring a sturdy location for the final structure.

Principles of FEA

Finite element analysis are based on principles that include boundary conditions, such as forces and pressures, as well as three governing equations:

1. Equilibrium equations, which find when the opposing forces or influences are balanced.

2. Strain-displacement relations, which measure the deformation that the design experiences under any given external impacts.

3. Constitutive equations, which are relations between two physical quantities, specific to the given metal or substance, which predict the material's response to external stimuli.

How FEA works

For finite element analysis to perform its necessary simulations, a mesh -- containing millions of small elements that together form the shape of a structure -- must be created. Calculations must be performed on every single element; the combination of each of these individual answers provides the final result for the full structure.

This process can be further broken into three steps: the preprocess, process and postprocess.

During the preprocess step, the user is asked to select the analysis type -- such as modal analysis or structural static analysis -- as well as the element type. Next, the material properties

must be defined and nodes must be made. The elements are then built by assigning connectivity at the nodes. Finally, boundary conditions and loads are applied. The computer performs the second step, the process. During this step, the computer solves the boundary value problem and then presents the results to the user.

During the postprocess step, the user reviews the generated results and notes factors such as:

➤ Displacement

➤ Temperature

➤ Time history

➤ Stress

➤ Strain

➤ Natural frequency

Designers using finite element analysis should be aware of inherent errors that can be found in this process, such as the simplification of geometry in the finite element method and use of basic integration techniques; errors in computing stemming from numerical difficulties or the limited number of digits available in computers; and common user mistakes, such as selecting the wrong type of element or providing inconsistent units of measurement.

Advantages of FEA

Finite element analysis provides the safe simulation of potentially dangerous or destructive load conditions and failure modes, allowing engineers to discover a system's physical response at any location. Other benefits include:

- Increased accuracy due to the analysis of any physical stress that might affect the design.

- Improved design because developers can observe how stresses within one element will affect the materials in another connected element.

- Earlier testing in the development process. Virtual prototyping allows the designer to model various designs and materials in a matter of hours, rather than the days or weeks it takes to produce hard prototypes.

- Increased productivity and revenue because FEA software allows developers to produce higher quality products in a shorter design cycle while also using less material.

- Enhanced insight into critical design parameters as the result of being able to model both the design's interior and the exterior. This allows designers to determine how critical factors affect the structure as a whole, as well as why and where failures might occur.

- Optimized use of models because one common model can be used to test several failure modes or physical events

- Fast calculation times and relatively low investment costs.

- Access to existing experimental results, which can be pulled from the parametric analyses of already validated models and applied to the new model.

Common applications of FEA

FEA is commonly used in mechanical, aerospace, automotive and civil engineering projects as well as biomechanics. Specifically, it is important for designing machines, analyzing fatigue for machines and their parts, certifying load capacities for lifting cranes, building airport bridges and determining brake or rotor lifetime certifications.

Other uses of finite element analysis include:

- Improving product safety

- Testing possible real world conditions on the design

- Reducing design and manufacturing costs

- Evaluating and optimizing alternative designs and materials

- Analyzing different basic solutions quickly

- Gaining product and company recognition and credit from locally and globally recognized certification authorities

- Optiming operations on workstations and personal computers

Other examples of FEA

Other common examples of finite element analysis include finding performance improvements for new and existing products; ensuring compliance with legal and contractual requirements; and defining the useful life of a product and establishing servicing schedules. FEA is also a commonly integrated tool in computer-aided design (CAD) packages.