Click Here

Browser does not support script. Updated: May 03, 2023 08:25 AM ESTFinite Element Analysis, or FEA, is the simulation of a physical phenomenon using a numerical mathematic technique called the Finite Element Method, or FEM. This process is at the core of mechanical engineering, as well as a variety of other disciplines. It is also one of the key principles used in developing simulation software. Engineers can use these FEM to reduce the number of physical prototypes and run virtual experiments to optimize their designs. Complex mathematics is required to understand the physical phenomena around us. These include things like fluid dynamics, wave propagation, and thermal analysis. Analyzing most of these phenomena can be done using partial differential equations, but in complex situations where multiple highly variable equations are needed, Finite Element Analysis is the leading mathematical technique. The history of finite element analysis The beginnings of FEA date back to the famous mathematician Euler in the 16th century. However, a more rigid definition of "FEA" traces the first mention of the method back to the works of Schellbach in 1851. Finite Element Analysis was a process developed for engineers by engineers to address structural mechanics problems in civil engineering and aerospace. Source: Craig Bonsignore/Flickr This practical intention of the methodology meant that these methods were designed as more than just mathematical theory from the beginning. By the mid-1950s, the techniques of FEA had become advanced enough that engineers could start using it in real-world situations. The mathematical principles of FEA are also useful in other areas, such as computational fluid dynamics or CFD. The key difference here is that FEA focuses on structural analysis, and CFD on fluid dynamics. What does running FEA entail? Essentially, FEA algorithms are integrated into simulation software like Autodesk Inventor Nastran or ANSYS's software suite. These programs are usually integrated into computer-aided design (CAD) software, making it much easier for engineers to go from design to running complex structural analysis. To run an FEA simulation, a mesh is first generated, containing millions of small elements that make up the overall shape. This is a way of transcribing a 3D object into a series of mathematical points that can then be analyzed. The density of this mesh can be altered based on how complex or simple a simulation is needed. Calculations are run for every element or point of the mesh and then combined to make up the overall final result for the structure. Since the calculations are done on a mesh rather than the entirety of a physical object, some interpolation needs to occur between the points. These approximations are usually within the bounds of what's needed. The points of the mesh where the data is known mathematically are referred to as nodal points and tend to be grouped around boundaries or other areas of change in an object's design. FEA can also be applied to thermal analysis within a material or shape. For example, if you know the temperature at one point in an object, how would you determine the exact temperature at other points of the object, dependent upon time? Utilizing FEA, an approximation can be made for these points using different accuracy modes. There's a square approximation, a polynomial approximation, and a discrete approximation. Each of these techniques increases in accuracy and complexity. If you're interested in the intense mathematical side of FEA, take a look at this post from SimScale that goes into the nitty-gritty. Computational fluid dynamics The other type of FEA we mentioned earlier is Computational Fluid Dynamics, which warrants a look into its use. The core of CFD is based on the Navier-Stokes equations, which examine single-phase fluid flows. In the early 1930s, scientists and engineers were already using these equations to solve fluid problems, but due to the lack of computing power, the equations were simplified and reduced to 2 dimensions. While rudimentary, these first practical applications of fluid dynamic analysis gave way to what would soon be an essential simulation asset. For most of the early years, solving CFD problems entailed simplifying equations to the point that they could be done by hand. By no means was the average engineer using these calculations; rather, up until the late 1950s, CFD remained a largely theoretical and exploratory practice. As you could probably have guessed, computing technology improved in the 1950s, allowing the development of algorithms for practical CFD. A team at the Los Alamos National Lab developed the first functional CFD computer simulation model in 1957. The team spent ten years working on these computational methods, which created the early models for much of the foundation of modern programs, spanning the vorticity-in-stream function to particle-in-cell analysis. By 1967, Douglas Aircraft had developed a working, 3-dimensional CFD analysis method. The analysis was fairly basic and was developed for fluid flow over airfoils. It later became known as the 'panel method,' as the analyzed geometry was largely simplified to simplify computation. From this point onward, the history of CFD is largely based on innovations in mathematics and computer programming. Full potential equations were incorporated into the methodology by Boeing in the 1970s. The Euler equations for transonic flows were incorporated into codes in 1981. While the early history of CFD is ripe with development, the companies involved in pursuing the technology were also notable. NASA and Boeing were the two key players in advancing computational techniques for CFD. By the 1990s, however, the technology and computing ability had become advanced enough that automakers also began seeing the application of CFD in automotive design. GM and Ford adopted the technology in 1995 and began making cars that were much more aerodynamic than the boxy wagons of the past. The history of CFD is riddled with big names in the industry, all of which have developed CFD analysis into one of the biggest simulation tools available. Understanding the complex mathematics behind CFD isn't necessary for many modern engineers to run simulations. The tools are not only used by experts in fluid dynamics and mathematics but can now be accessed by the everyday engineer with virtually any skill level. I don't know about you, but having access to some of the most mathematically powerful simulation analysis software as just a common engineer is pretty cool. Together, FEA and CFD algorithms built-in to modern CAD tools give engineers access to what are essentially mathematical superpowers. Trevor English Trevor is a civil engineer (B.S.) by trade and an accomplished writer with a passion for inspiring everyone with new and exciting technologies. He is also a published children's book author and the producer for the YouTube channel Concerning Reality.Stay up-to-date on engineering, tech, space, and science news with The Blueprint.By clicking sign up, you confirm that you accept this site's Terms of Use and Privacy PolicyNewsscience by Jhalak SharmaMarch 8, 2025blog When implementing engineering designs, engineers use the numerical method of Finite Element Analysis (FEA) as their essential analysis tool. FEA is an analytical tool to predict product responses toward operational forces consisting of heat, vibration, fluid motion, and structural applications. Engineers conduct finite element analysis to probe product weaknesses before they implement design enhancements and manufacturing. This article explains what is finite element analysis is while detailing its structural and mechanical applications, finite element analysis in design, and effective learning approaches. The computational approach of Finite Element Analysis (FEA) divides complex systems into smaller components referred to as elements. Nodes connect these elements into a finite element analysis model, which stands as a representative structure model. The approach implements mathematical formulas that run computer simulations to display how objects respond when faced with multiple forces. Finite Element Analysis Simulation plays a critical role in engineering design by predicting structural performance before manufacturing. Most structural and mechanical engineers use the finite element analysis method to analyse stress, together with strain and material deformation. Design engineers employ finite element analysis as a virtual prototype testing tool, which cuts down the requirement for physical tests. In the automotive, aerospace, construction, and biomedical engineering industries, FEA ensures safety while providing efficiency and cost-effectiveness. The finite element analysis method performs structured simulation procedures for accurate outcomes. Engineering teams conduct design testing using finite element analysis models before manufacturing. Preprocessing stands as the first step before the solution takes place, followed by postprocessing. Finite element analysis begins with preprocessing steps that form its essential base in the design process. Engineers develop a digital model that depicts the product as well as specifying its structural contours and substance characteristics. The analytical model requires meshing to segment its components into smaller units. The precision of finite element analysis results depends on selecting appropriate 1D, 2D, and 3D element types. Every mesh element requires specific property assignments, including its elasticity along with its density and thermal conductivity factors. At this point, engineers introduce heat as well as forces and pressure onto elements throughout the finite element analysis model. Simulation of real-world conditions happens through the deployment of boundary conditions that impose fixed supports and movement restrictions. FEA software computes stress and strain results as well as displacement outcomes by solving mathematical equations. Simulation results are subjected to post-processing analytical procedures. Through their analysis, engineers check for patterns of deformation together with pressure distribution and safety levels. Colour-coded stress maps exemplify how finite element analysis displays test results for evaluation purposes. The design verification process inspects whether the product satisfies both industrial quality benchmarks and necessary safety prerequisites. FEA Training Online Course helps you to get trained on Finite Element Analysis from basic concepts to advanced simulation analysis techniques. FEA experts offered live hands-on training on it. Engineers rely on correct finite element analysis (FEA) results for making essential engineering choices. Structural predictions under different conditions rely on finite element analysis models, which engineers utilize for their work. Design failures and manufacturing costs increase dramatically due to small simulation errors. An FEA simulation's accuracy depends directly on the quality of its mesh. Engineering results improve with increasing mesh density, yet insufficient meshing leads to flawed strain distribution. Engineers conduct mesh refinement procedures for precise calculations because improper results can occur due to ambiguous outcomes. Boundary condition implementation plays a vital role in achieving accurate results during finite element analysis in design. The application of inappropriate constraints generates false results that reduce product reliability. Engineers need to define their design parameters precisely to achieve better accuracy. The success of FEA simulations depends on material properties that operators input accurately. Defining material elasticity and thermal abilities, and overall strength, leads to flawed prediction outcomes. Finite element analysis models need proper definitions to guarantee real-world simulation accuracy. Different element types for FEA determine the accuracy levels. Design complexity requires various element dimensions from 1D and 2D up to 3D. When choosing elements that are not appropriate for each simulation model, they might generate inaccurate results. Read more: Which are the best FEA software: ABAQUS or ANSYS ? FEA stands as a fundamental method that designers use for product development processes. The application of this technique enables safer designs combined with improved performance as well as cost reduction. By performing FEA, engineers discover the material's reactions to different forces, such as tension and compression, as well as bending forces. A properly designed finite element analysis model delivers precise stress and strain evaluation. It also stops structural failure from happening. Through the finite element analysis method, designers achieve strength retention when they decrease material quantities. Strong, lightweight materials that benefit fuel efficiency and performance represent essential goals in the automotive and aerospace industries. The analysis method uses FEA to measure both fluid dynamics behaviour and heat distribution patterns. Aerospace and electronic industries utilise FEA to optimise cooling system designs, which enhances system efficiency while preventing overheating. Engineers use a proper selection of finite element analysis elements to verify that products satisfy safety requirements. Medical device industries apply FEA for implant and prosthetic durability testing, which ensures regulatory compliance. FEA is a vital design tool that allows engineers to enhance product designs before manufacturing. Engineers benefit from this tool by developing better designs at reduced expenses and increased performance outcomes. A company achieves structural optimisation through the finite element analysis without expensive prototypes. Through FEA, companies cut down their manufacturing costs by eliminating the necessity of physical testing. Through the finite element analysis method, engineers create virtual simulations that remove physical testing and avoid expensive mistakes in design. The design process using FEA leads to the acceleration of product development speed. Engineers utilise simulation to evaluate several product designs before final production begins. The company's ability to market new products accelerates because of this process. Applying finite element analysis explained techniques helps engineers locate product weaknesses at an initial stage. The system identifies potential failures in real-world applications, thereby creating safer products. Structural integrity depends on FEA because it evaluates different conditions against finite element analysis element types. The efficiency of a design process increases through FEA because it optimises material distribution and structural integrity strength levels. Product durability improves while performance reliability increases for items operating under environmental conditions and external loads. A physical structure becomes present in the finite element analysis model through its nodes and elements. Engineers apply this model for the study of various design situations to enhance product performance. Analysts utilize the finite element analysis method to make predictions about product stress, strain, and deformations before beginning manufacturing. The application of FEA includes evaluations in numerous domains. Automotive crash testing simulations – Ensures vehicle safety and impact resistance. The analysis of bridge and building structures checks their stability when tested under various loads. Aerospace component thermal evaluations – Test materials for heat resistance in extreme conditions. Biomedical implants and prosthetics testing – Ensures durability and compatibility with the human body. Finite Element Analysis (FEA) is a vital instrument that engineers and designers utilize daily. FEA benefits engineers by predicting and enhancing product performance. Knowledge of FEA enables professionals to perform efficient evaluations of structures and mechanical systems. Expertise in finite element analysis fundamental concepts enables professionals to thrive in their workplace. For professional training in FEA, reach out to PIGSO LEARNING to boost your engineering skills. We offer focused training for students and professionals, giving them skills relevant to the industry to excel in their professional lives. FEA stands for a process that examines structural responses against forces and heat conditions, along with other factors. Engineers break complex objects into multiple parts to achieve better performance analysis through this method. The FEA serves multiple industries, including automotive, aerospace, construction, and healthcare. Engineers deploy FEA to conduct car crash safety tests and design aircraft frameworks while developing resilient structures and medical equipment. The evaluation process ensures products achieve safety standards and operational efficiency before manufacturing. FEA produces accurate results only after creating an appropriate and well-built model. Better mesh elements paired with precise material characteristics and proper boundary constraints lead to more accurate results. Engineers optimize these factors to achieve specific outcomes. FEA operates with three element types. Beams (1D) – For long, thin structures like bridges. Shells (2D) – For surfaces like car bodies. Solids (3D) – For solid objects like engine parts. The selection process for elements is based on how sophisticated the analyzed structure is. You should begin learning FEA by taking educational resources from online courses, articles, and books. After learning basic software skills in any FEA Software Applications like Abaqus or ANSYS, move on to real-world project assignments. Students learn to solve problems and find solutions more effectively through practical application. I'm a passionate content writer who crafts compelling, audience-focused content. I specialize in turning ideas into engaging words, ensuring every piece delivers value, connects, and drives results. Finite Element Method (FEM) is a computational technique widely used in structural analysis to model and evaluate complex structures. FEM allows engineers to simulate how structures will respond to different loading conditions, including stress, strain, and displacement. In this article, we will explore the basics of FEM in structural analysis, its mathematical foundation, its types, and the process of applying it. Additionally, we will touch on the benefits and limitations of FEM in structural evaluation. Understanding the Basics of FEM Finite Element Method (FEM) is a numerical approach that divides a complex structure into smaller, simpler parts called finite elements. A set of equations governs these elements based on physical laws. By solving these equations, FEM helps engineers approximate the behavior of the entire structure. The goal is to predict how forces, pressures, and displacements affect it. One of FEM's primary strengths is its ability to handle complex geometries, materials, and boundary conditions, making it a versatile tool for a wide range of engineering applications. FEM works by creating a finite element mesh, a network of smaller parts that helps simulate and analyze local effects and their impact on the overall structure. Software like SDC Verifier further simplify this process by integrating automatic compliance checks with industry standards, reducing the risk of manual errors in structural verification. For further insight, watch this discussion by mathematician Gilbert Strang from MIT, as he delves into the origins of FEM and its evolution, as well as the challenges in applying it to fluid and gas problems. The Role of FEM in Structural Analysis FEM plays a vital role in structural analysis by transforming a continuous structure into a set of discrete elements, allowing for the detailed study of each part. Engineers use FEM to predict how a structure will respond to various load conditions, whether static or dynamic forces. The image below showcases the practical application of FEM, with a meshed component analyzed for stress distribution, visualized through color-coded stress gradients. Image: jousefmurad.com By simplifying complex structures into manageable elements, engineers can make precise predictions about how loads will affect the structure, ensuring stability and safety. FEM is particularly effective in handling real-world scenarios that involve curved material behavior or complex boundary conditions, such as bridges, high-rise buildings, or automotive crash simulations. Mesh generation within FEM enables engineers to evaluate stress, strain, and failure points within a structure, optimizing designs for safety and efficiency. Software solutions like SDC Verifier enhance this process by providing engineers with comprehensive reporting tools that align with industry standards, streamlining the analysis workflow. The Mathematical Foundation of FEM Principle of Minimum Potential Energy At the core of FEM is the Principle of Minimum Potential Energy. This principle states that a structure is in equilibrium when its total potential energy is minimized. When a structure deforms due to applied loads, it stores potential energy. FEM applies this principle by minimizing the stored energy in each finite element, which allows engineers to predict how the structure will behave under various loading conditions. The diagram below illustrates the relationship between work, force, and displacement in mechanical systems, which is fundamental to understanding energy principles in FEM. Image: researchgate.net Discretization in FEM Discretization is a fundamental step in FEM, where the continuous system (structure) is divided into finite elements. Each element has its own geometry, material properties, and relationships with neighboring elements. By discretizing the structure, engineers transform the problem of analyzing the entire system into solving a set of smaller, interconnected problems. The accuracy of FEM depends heavily on the type and size of the finite elements used. Engineers must carefully select element shapes (e.g., triangles, tetrahedrons) and sizes to balance computational efficiency with the precision of results. Proper meshing is crucial for capturing the correct behavior of the structure and minimizing errors in simulation. Different Types of FEM Linear FEM Linear FEM assumes that the relationship between applied loads and resulting displacements is proportional, meaning the structure's response is elastic and follows Hooke's Law. This method is commonly used when the structure's behavior remains within the elastic range, where stress and strain are directly proportional. Engineers often apply linear FEM to simpler problems, such as beams, trusses, and frames, in fields like civil, mechanical, and aerospace engineering. Its advantage lies in the simplicity and speed of computation, making it efficient for many practical applications. However, linear FEM is limited when dealing with complex phenomena such as plasticity, large deformations, or material failure. Non-linear FEM When structures undergo large deformations, or when materials exhibit non-linear stress-strain behavior, non-linear FEM is required. This method is essential for accurately modeling real-world situations where the assumptions of linear FEM no longer apply, such as in crash simulations, buckling analysis, or the study of plastic deformation. Non-linear FEM, though more computationally intensive, provides a more realistic assessment of structural behavior under extreme conditions. Engineers must account for factors like material plasticity, creep, and contact between parts to ensure accurate results. Despite the increased complexity, non-linear FEM is critical for designing safer and more reliable structures in fields such as automotive safety and high-performance engineering. The Process of FEM in Structural Analysis Pre-processing Stage During the pre-processing stage, engineers define the structure's geometry, assign material properties, and generate the finite element mesh. This involves converting the physical model of the structure into a mathematical one that can be solved using FEM. Engineers also apply boundary conditions and loads to simulate real-world operating conditions. At this stage, engineers may refine the mesh to improve the accuracy of the analysis, especially in areas where stress concentrations are expected. This step helps ensure that the FEM results are reliable and converge to an accurate solution. Solution Stage In the solution stage, the system of equations governing the behavior of each element is solved iteratively using numerical methods. This step calculates displacements, stresses, and strains across the structure. The results offer a detailed understanding of how the structure responds to applied loads, providing crucial information for design and verification. Engineers may also conduct sensitivity analysis during this phase, assessing how changes in material properties, geometry, or boundary conditions affect the structural response. This allows them to fine-tune the design for improved performance. Post-processing Stage In the post-processing stage, the results of the FEM analysis are visualized and interpreted. Engineers use software to create visual representations of the structure's deformation, stress distribution, and critical areas. These visualizations help engineers assess whether the design meets safety standards and performance criteria. Post-processing also includes fatigue analysis, which evaluates the structure's durability under repeated loading conditions. By predicting potential failure points due to fatigue, engineers can make necessary design modifications to enhance the structure's longevity and reliability. Benefits and Limitations of FEM in Structural Analysis Advantages of Using FEM Accurate Predictions: FEM enables engineers to make highly accurate predictions about how structures will respond to different loads, ensuring safety and performance, especially in critical applications like aerospace and civil engineering. Design Optimization: The detailed insight into stress, strain, and failure modes allows engineers to optimize designs, improving safety while reducing overdesign and material costs. Virtual Prototyping: FEM reduces the need for costly physical prototypes by allowing engineers to test designs virtually under various load conditions, speeding up the design process and facilitating multiple iterations. Potential Drawbacks of FEM Cost and Complexity: FEM requires specialized software and hardware, which can be costly for smaller firms. Additionally, the complexity of the software may present a steep learning curve for engineers unfamiliar with the method. Accuracy Dependence: The accuracy of FEM results relies heavily on correct assumptions, material properties, and mesh quality. Poor meshing or incorrect material modeling can lead to errors in analysis, compromising the safety and integrity of the structure. Conclusion FEM is a powerful and versatile method for structural analysis, allowing engineers to simulate and evaluate complex systems with great precision. A strong understanding of FEM principles, from mesh generation to post-processing, is essential for optimizing designs and ensuring structural integrity. While FEM offers many advantages, its limitations, such as cost and complexity, must be carefully managed. By using advanced tools like SDC Verifier, which integrate FEM with automated code-checking and reporting, engineers can streamline their workflows and ensure compliance with industry standards. Structural Guide Scroll to Top Recent Articles Archive posts When designing a new product, having the ability to make prototypes is ideal as it means issues can be found early in the process. However, not all projects have this luxury so must be right first time – a challenge we're used to seeing. Many of the projects we deliver must be right first time, whilst incorporating challenging mechanisms and structures, plus a variety of materials. So how do we overcome this challenge? We use computer simulation to model load cases throughout all design and development projects. It's an iterative process to create concept designs, run FEA scenarios, then develop and optimise the design for maximum performance. Here's a quick explanation of the different types of analysis that can be utilised: Linear & Non-Linear FEA Materials can either be linear elastic (where the relationship between stress and strain in the material is linear – for example aluminium or steel), or non-linear elastic (where the relationship between stress and strain in the material is non-linear – for example, polymers). Linear elastic materials are typically linear elastic up to a point, and then start displaying non-linear elastic behaviour. Linear FEA is carried out when a part is made from a linear elastic material, and there is not much deformation caused by the forces being applied to the part. Where the material is non-linear elastic or the forces being applied can cause large displacements, a non-linear FEA must be performed. Static FEA: A linear static FEA is performed when the forces applied to a part or assembly do not change over time. A static FEA is the simplest form of FEA, and is performed exclusively on linear elastic materials, where small displacements are anticipated. Dynamic FEA: A dynamic analysis is performed when the forces applied to a part or assembly change over time. A dynamic analysis can be performed on either linear elastic or non-linear elastic materials. Common examples of a dynamic study include drop tests and vibration analysis. CFD – Computational Fluid Dynamics: CFD simulates how gasses or liquids flow in, or around, a part or assembly. Thermal Analysis: Thermal analysis simulates heat transfer between solid bodies. A thermal analysis can be a useful standalone tool, for example when analysing the design of a heat-sink. It can also used in combination with a static analysis to conduct thermal stress analyses on parts or assemblies. Using FEA can be hugely beneficial to a project as it can reduce costs, save time and improve product performance. Further benefits arise with lower environmental impact by using reduced material usage for optimised rigidity/flexibility/weight. Effective use of FEA throughout the process will ultimately ensure your design is fit for purpose. One of the biggest issues we find with FEA being outsourced, is that the iterative nature of design optimisation can be broken or come too late. The truth is that it's typical for analysis to identify issues in a design, which subsequently require further investigation and design optimisation. Therefore, it is always an advantage to introduce FEA, as early as possible, into the design process. This means problems are identified more quickly and changes can be made to the basic design early on. Doing this well before extensive engineering is undertaken will significantly reduce time and therefore costs or delays. If you need any advice or guidance on the effective use of FEA to speed up your product development process, please get in touch and we will be glad to help.