

YOUR COMPANY'S GUIDE TO **FINITE ELEMENT ANALYSIS**

Helping your organization understand
Finite Element Analysis as a tool for solving
complicated engineering problems.



ASME **L&D** LEARNING &
DEVELOPMENT

INSIDE THIS GUIDE

CONTENTS

Finite Element Analysis	3
When to use Finite Element Analysis	4
The basic steps of Finite Element Analysis	5
Benefits of ASME training	10
Introduction to Finite Element Analysis	12
Advanced Finite Element Analysis	13
More about our courses	14
Let's talk about it	15



FINITE ELEMENT ANALYSIS

Originally developed for aerospace structural analysis, Finite Element Analysis (FEA) can help engineers simulate real-world conditions and find vulnerabilities in their design prototypes.

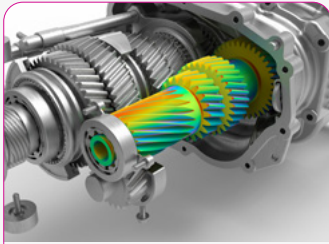
Using the finite element method, engineers divide complex geometry into simpler and more manageable parts called elements. The behavior of these elements are then approximated, and the results combined to predict the outcome for the entire structure.

FEA is designed to forego the need for expensive and lengthy testing and prototyping by simulating the performance of components before they are built using software. FEA can expedite the design and engineering process, and ultimately improve the integrity of any given structure because this method allows engineers to study the behavior of a product or structure under several different conditions without having to build multiple physical prototypes.



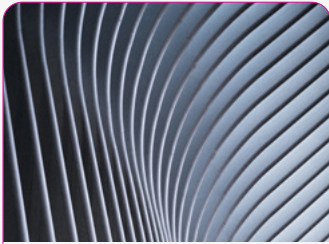
WHEN TO USE **FINITE ELEMENT ANALYSIS**

FEA can be used in many situations — it's particularly useful when the physical testing of a structure is too time-consuming and costly to undertake. A few examples of when an engineering professional may elect to use FEA include:



Failure analysis

To discover and analyze the root cause of a component failure and manufacturing defects.



Material selection

To compare the performance and suitability of different materials for a structure.



Regulatory compliance

To demonstrate compliance with certain regulations like safety and performance.



THE BASIC STEPS OF FINITE ELEMENT ANALYSIS

There are six steps to Finite Element Analysis — all (aside from the first step in some cases) of which are necessary for successful structural analysis. By following the six steps, engineers can be sure that there's proper definition, accurate modeling and appropriate validation of the structure or component in question.

Step one:

Definition of the solution field.

Step two:

Discretization.

Step three:

Development of the finite element equations.

Step four:

Adoption of loads and boundary conditions.

Step five:

Solution for primary response variables.

Step six:

Post-processing.

THE BASIC STEPS OF FINITE ELEMENT ANALYSIS

Step one:

Define the problem

This step isn't always necessary — simple models can be built by going directly to the discretization process. For more complicated models, definition of the solution field is a simplification step.

Commercial software packages use a frontend user interface to define the shape of the field as a solid model, which is then replaced in the next step of analysis with a finite element model.

The solid model is made up of points, lines, areas and volumes, serving only to define the shape of the field. Whereas the finite element model consists of nodes and elements, and contains all the necessary information for solving the problem.



Step two:

Discretize the structure

A sufficiently large but finite set of points is chosen, representing locations within the field. These points (the nodes) are then used to define the boundaries of discrete subregions of the field (the elements). These elements combine to approximate the shape of the field and will, in the end, provide the approximate equations to be solved for the primary response variables (also known as the field variables).

Note on the term 'finite':

The term “finite element” reflects both the fact that the element is finitely sized (neither grossly small nor overly large), and possesses a finite number of DOF. This is a useful point to remember, in order to understand the philosophy of the method.

THE BASIC STEPS OF FINITE ELEMENT ANALYSIS

Step three:

Formulate element equations

Depending on the nature of the problem, there are a variety of ways to generate the elemental equation set:

- The Direct Method uses the known equations of, for example, strength of materials to generate element equations. This is only possible for simple problems.
- Functional Minimization is possible for indirect generation of equations. This requires a function to be minimized (as, for example, the principle of minimum total potential energy). In practice, it is often possible to recast a differential equation into an equivalent functional form to be minimized.
- Methods of Weighted Residuals may be used to develop an element for approximate solution of a differential equation, without development of a functional form.

Example:

The elemental equations take the form: $\mathbf{k}\mathbf{d} = \mathbf{f}$

$\mathbf{d}$ is a vector of degrees of freedom of the nodes

$\mathbf{k}$ is called the “stiffness matrix”

$\mathbf{f}$ is the vector of nodal “forces”

We refer to $\mathbf{k}$ as the “stiffness” matrix. In such a case, $\mathbf{f}$ is a vector of applied THERMAL loading.

Once the element equations are developed, the global set is assembled in such a way as to express interconnectedness of nodes. The elements share nodes, so that the field contains no inappropriate gaps or overlaps.

The resulting equations are algebraic, in the form: $\mathbf{K}\mathbf{D} = \mathbf{F}$

where components of $\mathbf{D}$ and $\mathbf{F}$ are to be found in the solution.

THE BASIC STEPS OF FINITE ELEMENT ANALYSIS

Step four:

Assemble loads and boundary conditions

The global FE equations, $\mathbf{KD} = \mathbf{F}$ are not solvable with the exception of eigenproblems (for example, buckling and free vibration).

Even if the values of D are sufficiently known, we can solve the global equations only if we know enough terms from the vector of forces, $\mathbf{F}$. Thus, the application of both loads and boundary conditions is an essential part of finite element analysis.

The actual nature of the loads and boundary conditions which can be used will depend on the element type chosen for the analysis, and the problem to be solved.

Example:

In a thermal problem, the same idea yields that we must sufficiently constrain temperature, so that the final temperature field is forced to take an unambiguous value. For example, it would not be possible to find “the” temperature distribution across a wall with convection on one side (of known bulk temperature and film coefficient), without knowing the temperature somewhere in or on the wall, as a boundary condition.

THE BASIC STEPS OF FINITE ELEMENT ANALYSIS

Step five:

Solve the equations

After application of loads and boundary conditions, a set of algebraic equations emerges, which has a non-singular stiffness matrix and known force vector. This set is solvable using standard mathematical or computational tools. The result is the set of all of the unknown nodal degrees of freedom.

Supplementing this set with the known nodal DOF (i.e. the boundary conditions) yields the set of all nodal DOF.



Step six:

Evaluate the results

The field variable has, by now, been approximated at all nodal locations. The final step in the computational portion of FEA is generation of secondary (higher order) variables. In a structural problem, FEA solves for displacements.

Post-processing allows solutions for unknown forces, strains, stresses. In other problems, the secondary variables are other things.

KEY BENEFITS OF ASME TRAINING FOR **FINITE ELEMENT ANALYSIS**



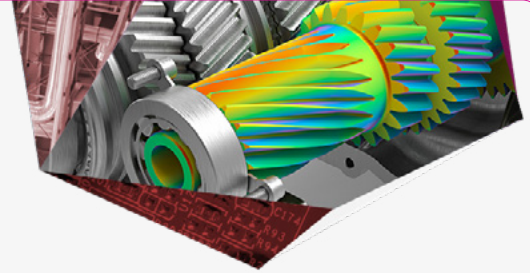
Training of personnel

Every organization that takes part in FEA training benefits from gaining a deeper understanding of the FEA fundamentals, software and techniques, which they can then apply throughout their work.



Better collaboration

Corporate training for FEA empowers employees to work better together in their problem solving as they develop common skills in applying FEA to their designs.

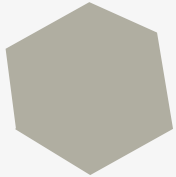


Quality assurance

With a better understanding of FEA, engineering professionals can create better and more reliable designs that not only meet safety requirements, but also reflect first-class design industry standards.

FREQUENTLY ASKED QUESTIONS

We asked our expert Scott Steinbrink some of our most frequently asked questions about FEA.



Scott Steinbrink

Scott Steinbrink holds a PhD in Engineering Mechanics from Virginia Tech, and currently teaches Mechanical Engineering at Gannon University, Erie, PA. At Gannon, Scott specializes in design and solid mechanics, for each of which Finite Element Analysis is an essential tool. He also teaches (for the past 25 years) the introductory and advanced FEA courses at Gannon.

When should I be using Finite Element Analysis, and when is it not appropriate?

Finite element analysis can be used at any stage of product development. An FE model can be used to predict in-service performance during the design phase and, in some instances, to predict effectiveness of the manufacturing phase, and can be used to predict performance of the final designed object.

Post-design, finite element analysis is appropriate in cases where physical prototype testing is overly costly, cumbersome or time consuming. A properly verified finite element model can in many cases serve as a surrogate for a physical model, allowing for low-cost and rapid design validation.

Of course, finite element analysis is a “tool in the bag,” so to speak, and can never replace engineering judgment; it is an approximation technique that is as good as the information provided to the computer and thus to be used cautiously in fields of knowledge that are not well understood by the FE analyst. It supplements – not replaces – the input of experienced engineers and technologists.

In what ways does Finite Element Analysis help me to plan ahead during the design stage?

Incorporated into most commercial FE applications is the ability to optimize designs and/or perform design sensitivity analyses. Given a preliminary design, the software can predict the performance of that design under assumed service conditions and provide valuable feedback with respect to its capabilities. Faults and inefficiencies can then be identified and rectified. Many models can easily and quickly be created as variations on the initial theme, including changes of material, geometry or load states, and can be submitted and assessed side-by-side. The result can provide very valuable feedback with respect to likely fruitful paths for improvement.

INTRODUCTION TO FINITE ELEMENT ANALYSIS

This course explains introductory FEA concepts underlying the creation of elements to make accurate approximations.

Topics covered

Module 1:	Introduction.
Module 2:	Details of the method.
Module 3:	Introduction to Abaqus.
Module 4:	More advanced topics in element generation.
Module 5:	Additional Abaqus capabilities.
Module 6:	Practical advice for competent FEA.

You will learn to:

- Provide examples of all the steps necessary to conduct a successful FEA from start to finish.
- Explain the concepts underlying the creation of elements which are used to make accurate approximations.
- Use finite element software for more advanced structural, thermal analysis and basic modal analysis.

ADVANCED FINITE ELEMENT ANALYSIS

Based on practical application of Abaqus software, this course builds on the introductory level course to provide a fuller understanding of how Abaqus works, as well as FEA in general.

Topics covered

Module 1:	Abaqus Command Line Inputs.
Module 2:	Nonlinear FEA.
Module 3:	Contact problems in Abaqus.
Module 4:	Structural Dynamics.
Module 5:	Additional Capabilities of FEA, with reference to Abaqus.

You will learn to:

- Identify command-line input for Abaqus.
- Describe structural dynamics, including modal and harmonic response analyses.
- Identify Structural Dynamics II, featuring transient dynamic analysis.
- Explain nonlinear structural analyses.
- Demonstrate design optimization in Abaqus.

MORE DETAILS ABOUT OUR COURSES

ASME is a world-class provider of mechanical engineering courses committed to delivering accredited, high-quality, trustworthy training to you and your team—backed by a globally recognized independent third party, IACET.

ASME Learning & Development provides technical and management courses to teach engineering codes, standards and best practices to individuals and teams across a range of disciplines.

Courses are designed for engineers of all levels from early career engineers to experienced engineers. And, thanks to the reopening of our sites, our courses can be accessed either virtually or in-person – with the ability to tailor content to include specific use cases and pain points for corporate training customers with eight or more learners.

LET'S TALK ABOUT IT

ASME is the go-to expert in both fundamentals and applications of Finite Element Analysis, and our courses are designed to help professionals remain consistent in their usage throughout the engineering process.

All ASME courses include real-world examples so that learnings can be applied to projects that your company is working on. With ASME Corporate Training, your team can address your unique challenges with an expert in a private and confidential setting.

Get in touch today to inquire about ASME Corporate Training.

► learningsolutions@asme.org

► go.asme.org/evolve