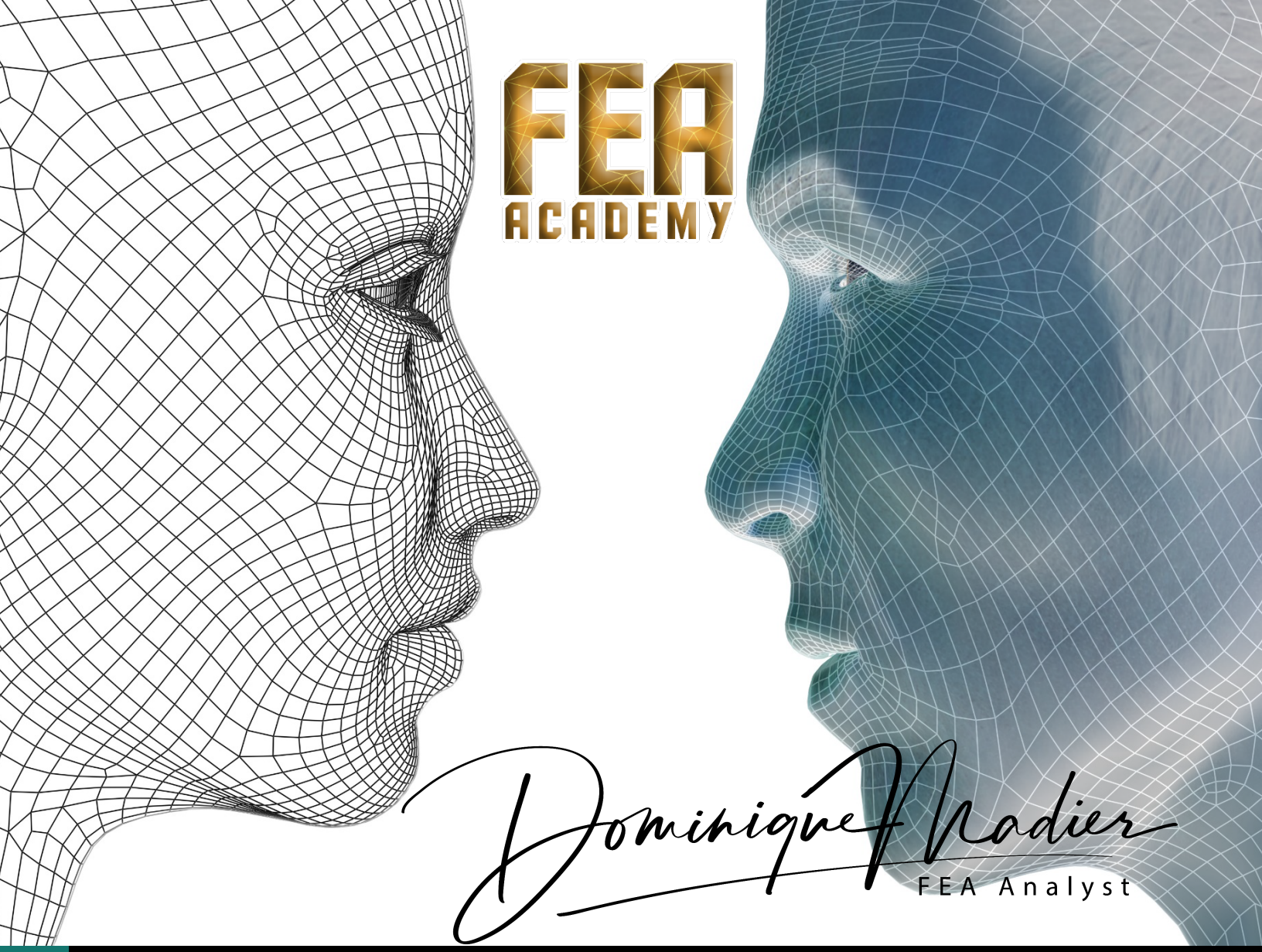



# An Introduction to the Fundamentals of Mesh Generation in Finite Element Analysis

At the heart of FEA lies the process of mesh generation, a critical step that can significantly influence the accuracy, efficiency, and success of a simulation. This article summarizes the stage for an in-depth exploration of mesh generation.

By Dominique Madier

© 2023 FEA Academy

Published by FEA Academy
www.fea-academy.com
Contact the author at: dominique.madier@fea-academy.com

© 2023 FEA Academy. All Rights Reserved.

ISBN: 978-1-9990475-3-5

**Trademarks**

Commercial software names, company names, and trademarks contained in this e-book are used only for identification and explanation without intent to infringe on their property rights.

**Legal Disclaimer and Exclusion of Liability**

The author reserves the right to make changes to information contained in this e-book, without prior notice. The concepts, methods, and examples presented in this text are for illustrative and educational purposes only and are not intended to be exhaustive or to apply to any particular engineering problem. The author assumes no liability or responsibility to any person or company for direct or indirect damages resulting from the use of any information contained herein.

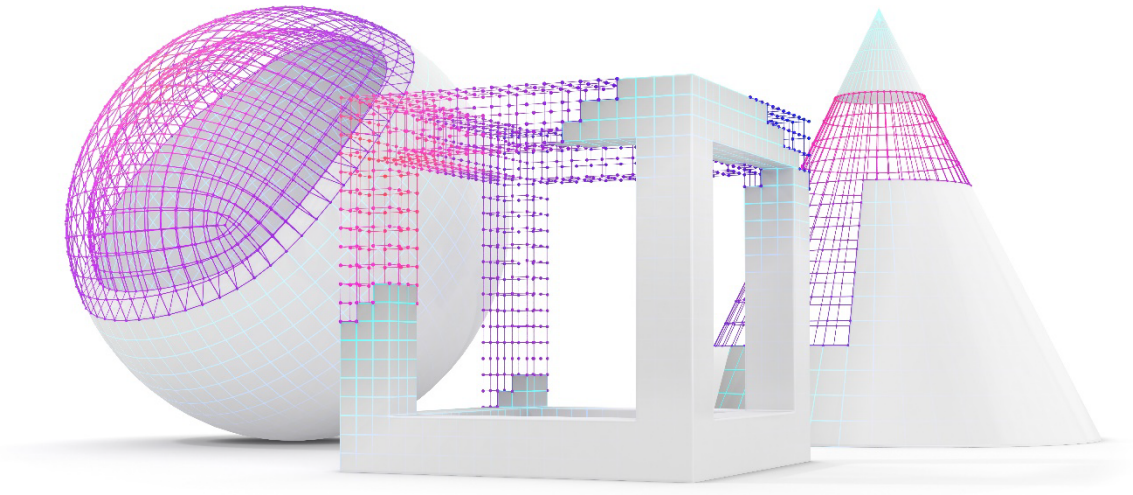
**Intellectual Property Rights**

Any reproduction of any extract of this e-book by any means whatsoever, including photocopying, microfilm, or scanning, is strictly prohibited without the written permission of the author. Unauthorized reproduction of this publication will be considered as a copyright infringement.



© 2023 – FEA ACADEMY

MESHING IS WHERE SCIENCE MEETS INTUITION IN FEA.
IT REQUIRES AN UNDERSTANDING OF THE PHYSICS INVOLVED
AND A FEEL FOR THE BEHAVIOR OF THE STRUCTURE UNDER STUDY.





© 2023 – FEA ACADEMY

# Table of Contents

**INTRODUCTION** ..... **8**

**ABOUT THE AUTHOR** ..... **10**

**ABOUT FEA ACADEMY** ..... **11**

**SECTION 1. UNDERSTANDING WHAT IS FEA** ..... **12**

What is Finite Element Analysis (FEA) ..... 12

From the Need for Analysis to the Need for Modeling ..... 16

The FEA Concept ..... 17

**SECTION 2. UNDERSTANDING MESH IN FEA** ..... **18**

What is Meshing? ..... 18

Definition and Purpose ..... 18

Process Overview ..... 19

Importance of Meshing in FEA ..... 19

Types of Elements ..... 20

1D Elements (Beam) ..... 20

2D Elements (Shell) ..... 20

3D Elements (Solid) ..... 21

Choosing the Right Element Type ..... 22

Examples of 1D, 2D and 3D Usage ..... 24

Required Dimensions for Each Type of Element ..... 25

How to Choose the Right Element ..... 26

Predict Your Structure's Behavior ..... 26

Experiment Your Library of Elements ..... 26

Understanding Element Behavior ..... 26

Rules for Selecting the Right Element ..... 27

Mesh Quality and its Impact on Analysis ..... 28



© 2023 – FEA ACADEMY

Defining Mesh Quality ..... 28
Impact on Results ..... 28
Balancing Mesh Quality and Computational Efficiency ..... 29

**SECTION 3. MESH GENERATION TECHNIQUES ..... 30**

Manual vs Automated Meshing ..... 30
Manual Meshing ..... 30
Automated Meshing ..... 31
2D Shell Meshing ..... 32
Overview of 2D Shell Meshing ..... 32
Advantages of 2D Shell Meshing ..... 32
Challenges and Considerations ..... 32
Tetrahedral vs Hexahedral 3D Meshing ..... 33
Tetrahedral Meshing ..... 33
Hexahedral Meshing ..... 34
Advanced Techniques: Sweeping, Paving, and Mapped Meshing ..... 35
Sweeping ..... 35
Paving ..... 37
Mapped Meshing ..... 37
Why Mesh in 2D Instead of 3D? ..... 38

**SECTION 4. MESH DENSITY AND ELEMENT SIZE ..... 39**

Determining Appropriate Mesh Density ..... 39
Understanding Mesh Density ..... 39
Strategies for Determining Mesh Density ..... 40
Importance of Element Size and Aspect Ratio ..... 40
Element Size Considerations ..... 40
Aspect Ratio and its Impact ..... 41
Balancing Accuracy and Computational Efficiency ..... 41
Achieving an Optimal Balance ..... 41



© 2023 – FEA ACADEMY

Techniques for Balancing ..... 42
Rules for Modeling Holes and Fillets ..... 43
**SECTION 5. MESH REFINEMENT TECHNIQUES ..... 45**
Local vs Global Mesh Refinement ..... 45
Global Mesh Refinement ..... 45
Local Mesh Refinement ..... 46
Adaptive Mesh Refinement (AMR) ..... 47
Concept and Advantages ..... 47
Implementation Challenges ..... 47
Practical Examples of Refinement Strategies ..... 48
Case Studies and Applications ..... 48
Best Practices and Guidelines ..... 49
Convergence Study Methodology ..... 49
Over What Distance is the Mesh Refined? ..... 51
**SECTION 6. COMMON PITFALLS IN MESH GENERATION ..... 52**
Over-refinement and Under-refinement Issues ..... 52
The Impact of Over-refinement ..... 52
Challenges of Under-refinement ..... 52
Inappropriate Element Choice ..... 53
Selecting the Wrong Element Type ..... 53
Avoiding Element Mismatch and Inconsistencies ..... 53
Ignoring Geometric Nonlinearities and Complexities ..... 53
Overlooking Geometric Nonlinearities ..... 53
Addressing Complex Geometries ..... 54
**SECTION 7. MESH VALIDATION AND ERROR CHECKING ..... 55**
Techniques for Mesh Quality Assessment ..... 55
Overview of Mesh Quality Metrics ..... 55
Tools for Mesh Quality Evaluation ..... 56



© 2023 – FEA ACADEMY

Error Estimation and Convergence Testing ..... 56
Understanding Error Estimation ..... 56
Importance of Convergence Testing ..... 56
Real-world Examples of Mesh Validation ..... 57
Case Studies in Industry ..... 57
Learning from Practical Applications ..... 57
**SECTION 8. GUIDELINES FOR EFFECTIVE MESH GENERATION ..... 58**
Step-by-Step Approach for New Projects ..... 58
Initial Assessment and Planning ..... 58
Geometry Preparation and Simplification ..... 58
Mesh Generation and Refinement ..... 58
Best Practices in Meshing for Different Types of Analysis ..... 59
Structural Analysis ..... 59
Thermal Analysis ..... 59
Fluid Dynamics ..... 59
Tips from Industry Experts ..... 60
Leveraging Experience and Case Studies ..... 60
Continuous Learning and Adaptation ..... 60
**CONCLUSION ..... 61**
Recap of Key Takeaways ..... 61
Essential Role of Meshing in FEA ..... 61
Adaptation to Different Analysis Needs ..... 61
Common Pitfalls and Best Practices ..... 61
The Future of Mesh Generation in FEA ..... 62
Emerging Technologies and Trends ..... 62
Integration with Design and Manufacturing ..... 62
Adapting to Complex and Multiphysics Analyses ..... 62
Meshless Solutions in Finite Element Analysis ..... 63
Introduction to Meshless Methods ..... 63



© 2023 – FEA ACADEMY

Advantages Over Traditional Meshing ..... 63
Challenges and Limitations ..... 63
Current Applications and Future Potential ..... 63
Integrating Meshless Solutions into Conventional FEA Practices ..... 63
Conclusion ..... 64



© 2023 – FEA ACADEMY

# Introduction

In the realm of engineering analysis, Finite Element Analysis (FEA) stands as a cornerstone, a powerful tool that enables engineers to predict how products will react to real-world forces, vibration, heat, fluid flow, and other physical effects.

At the heart of FEA lies *the process of mesh generation*, a critical step that can significantly influence the accuracy, efficiency, and success of a simulation. This article sets the stage for an in-depth exploration of mesh generation, a subject that, while often overshadowed by more glamorous aspects of FEA, is fundamental to its success.

Mesh generation in FEA is the art and science of discretizing a continuous domain into a finite number of elements, which can be effectively used to approximate the behavior of complex structures under various loads and constraints. The quality of the mesh directly impacts the precision of the simulation results, making it imperative for engineers and analysts to understand and master this process. However, mesh generation is not a one-size-fits-all solution; it requires a nuanced approach, tailored to the specific requirements of each analysis.

Throughout this article, we will delve into various aspects of mesh generation, including types of elements (1D, 2D, and 3D), meshing techniques, and the critical role of mesh density and element size. We will explore both manual and automated meshing methods, highlighting their advantages and limitations.

Furthermore, we will provide practical guidelines for mesh refinement, discuss the influence of material properties and boundary conditions on mesh design, and address common pitfalls to avoid during the meshing process.

Mesh generation, while a technical and often complex task, is integral to the success of FEA. Whether you are a seasoned analyst or a novice engineer, understanding the fundamentals of mesh generation is essential to harnessing the full power of FEA. This article aims to equip you with the knowledge and tools to create effective and efficient meshes, thereby enhancing the reliability and accuracy of your FEA results.



© 2023 – FEA ACADEMY

Join us as we embark on this journey through the critical world of mesh generation in finite element analysis.




Montreal – Canada – December 2023

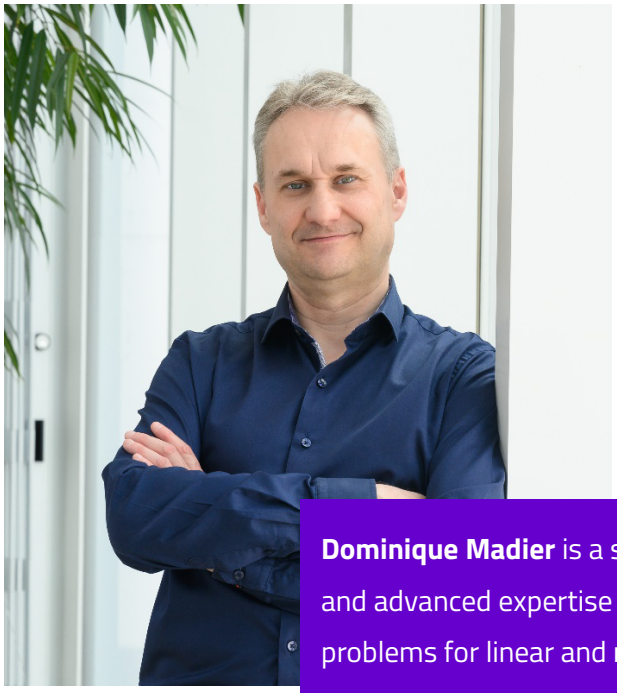
| FEA Academy Director |
| FEA Consultant | FEA Trainer | Book Author | Speaker |
| Helping Engineers and Students to Learn FEA |
dominique.madier@fea-academy.com
www.fea-academy.com






© 2023 – FEA ACADEMY

# About the Author



**Dominique Madier** is a senior aerospace consultant with 25 years' experience and advanced expertise in Finite Element Analysis (FEA) of static and dynamic problems for linear and nonlinear structural behaviors.

He has conducted detailed finite element analyses for aerospace companies in Europe and in North America (e.g., Airbus, Dassault Aviation, Hispano-Suiza [now Safran], Bell Helicopter Textron Canada, Bombardier Aerospace, Pratt & Whitney Canada, Beta Technologies and their subcontractors) on metallic and composite structures such as fuselages, wings, empennages, nacelles, engine pylons, helicopter airframes, and systems.

He is the author of the book "Practical Finite Element Analysis for Mechanical Engineers".

In 2020, he founded the FEA Academy in order to offer FEA consulting as well as online FEA courses for students and engineers.

He earned a Master's degree in Mechanical and Aerospace Engineering from Paul Sabatier University, Toulouse, France.



© 2023 – FEA ACADEMY

# About FEA Academy

FEA Academy is a company specializing in services related to numerical simulation and Finite Element Analysis, particularly in the realm of mechanical structural analysis. The scope of its services appears to encompass two main areas:

**1. Consulting Services in FEA:** FEA Academy offers expert consultation for projects that involve mechanical structural analysis. This likely involves providing insights and guidance on how to apply FEA methodologies effectively in various engineering projects. The consultation could cover aspects like model development, analysis strategies, interpreting FEA results, and optimizing designs based on FEA findings.

**2. Online On-Demand Practical Courses:** Apart from consultation, FEA Academy also focuses on education and training. It offers online courses targeted at students in engineering and professionals involved in structural projects. These courses are designed to be practical and application-oriented, providing learners with hands-on experience in FEA in structural analysis. This aspect of FEA Academy's offerings is particularly beneficial for enhancing the skills and knowledge of individuals who are new to FEA or those looking to deepen their expertise in this area.

Dominique Madier involvement likely ensures that the services and courses offered by FEA Academy are grounded in practical knowledge and are up to date with current industry practices.

www.fea-academy.com



© 2023 – FEA ACADEMY

# Section 1. Understanding What is FEA

## What is Finite Element Analysis (FEA)

For a given structural problem, the behavior of the structure can always be described by governing differential equations. To approximately predict the behavior of the structure, it is necessary to find a solution to the governing differential equations.

The fundamental concept of the finite element method is based on a discretization process that involves dividing the system to be analyzed into a finite number of subdomains called elements. Specific assumptions, referred to as shape functions, are applied across each element to represent the variation of the unknown dependent variables. The variation is computed in terms of solution values at specific locations called nodes. The method provides an algebraic system of equations for unknown nodal values that approximates the continuous solution.

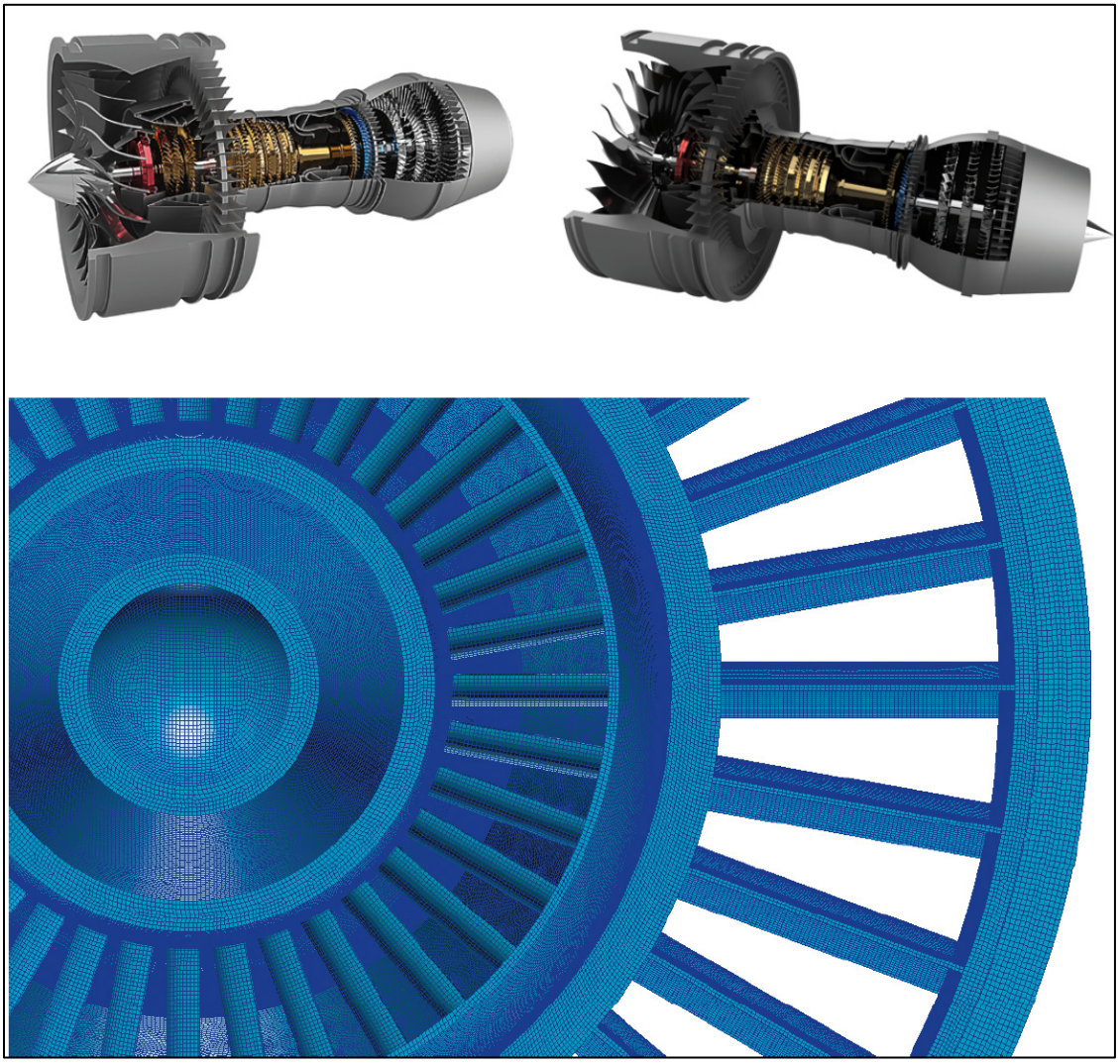
The finite element method is used to solve complex problems of engineering and is useful for problems with complicated geometries, loadings, and material properties, in which analytical solutions are not applicable. You can employ the finite element method in multiphysics problems to provide solutions that would otherwise be difficult to obtain. The method is usually employed to obtain approximate results for problems that are otherwise impossible to solve with classical methods.

Finite element analysis (FEA) is used to predict the response of mechanical structures to environmental factors such as forces, pressure, temperature, vibrations, and so on. With FEA, you can approximately predict how a product will behave in the real world, i.e., whether it is likely to break. You can also predict the weakest points of a product and verify whether it will work the way it was designed to.

For engineers, FEA is an important part of the product design phase in an iterative development process because it saves time and reduces the effort and cost associated with building physical prototypes. Prototype tests do not provide the type of numerical information that FEA does, and they make the product development process longer.



© 2023 – FEA ACADEMY



The aerospace industry, for example, still builds prototypes (called test cells) but, thanks to FEA, only prototypes representing the final configuration are required. During the product development process, simulation software that have FEA functionality are used to determine the deformations and stresses the components will undergo when subjected to environmental factors. These packages are extremely powerful when used intelligently and help to define the critical dimensions of a component by considering the factors of safety, weight, and other parameters.



© 2023 – FEA ACADEMY

FEA can be as simple as addressing a single part, for which you wish to capture the behavior, but it can also be more complex. Indeed, complex analysis may involve multiple parts for which you wish to determine the interactions.

These days, even the simplest products rely on FEA for design evaluation because contemporary design problems cannot be solved as accurately and cheaply with any other currently available method. Physical testing was the norm in years gone by, but now, it is simply too expensive. Most everyday products have been engineered using FEA; in fact, it is almost impossible to imagine a product designed without its help. The most common application, by far, is structural FEA, which determines how a structure responds to various applied loadings (forces, pressure, temperature, etc.).

FEA is useful at every stage of product design, as follows.

- Before, to define the product: FEA helps the engineer create a feasible design.
- After the product is designed: FEA permits the validation of the defined design.
- After the product is released: FEA can be used to investigate product failures.

To understand what FEA is, consider the meaning of the term "finite element analysis":

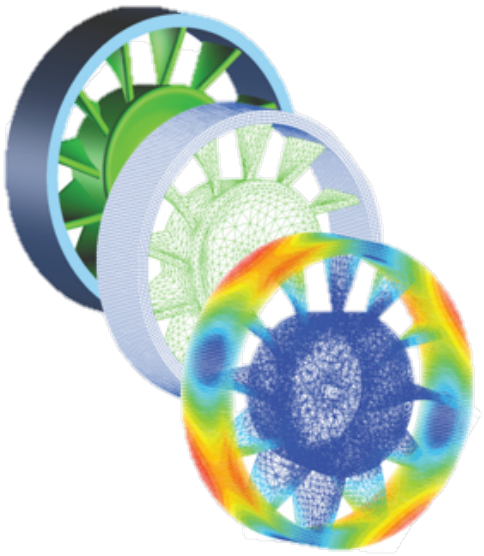
| FINITE | ELEMENT | ANALYSIS |
|---|---|---|
| Refers to the countable number of elements used to represent the structure. The elements are of finite and measurable size. | Refers to each small section of the analyzed part. An element acts on its neighboring elements.<br><br>The governing equations can be derived by considering a small section and writing equations for what is happening in that section. Then, the size of the section is mathematically allowed to become infinitesimal. | Means that the part is analyzed under certain conditions (static, dynamic, modal, buckling, linear, nonlinear...).<br><br>FEA assembles the equations from all the elements into one large matrix equation. The numerical solution of this large matrix equation is determined by a computer. |



© 2023 – FEA ACADEMY

FEA is based on the idea that if you divide a structure into multiple elements, the behavior of an individual element will provide a good approximation of local behaviors (stress and strain). Thus, the global assembly of such elements will reflect the behavior of the actual structure. While you may not be able to determine how a complex structural piece will behave, you can do so for a simple part of it. Therefore, if you assemble enough of these simple pieces, you can determine how a complex structural piece will behave.

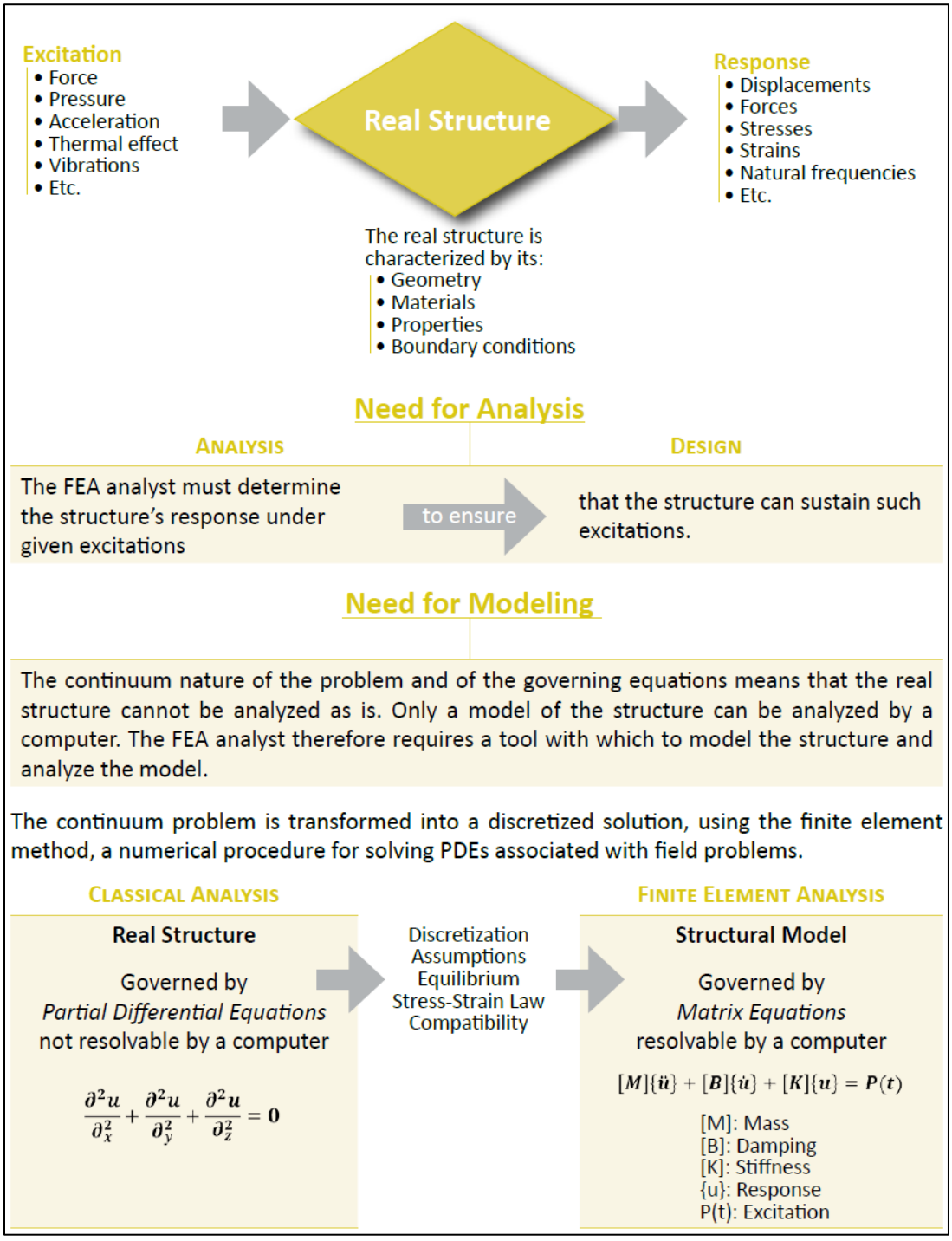
PDEs are used to formulate physical problems involving functions of several variables and can describe a wide variety of phenomena such as elasticity, acoustics, heat, electrostatics, electrodynamics, fluid dynamics, quantum mechanics, and even economy. Just as ordinary differential equations model one-dimensional systems, PDEs often model multidimensional systems. Thus, in the finite element method, a continuous system is idealized by a discrete system called the mesh. A structural problem with an infinite number of degrees of freedom (a continuous system governed by PDEs) is converted into a problem with a finite number of degrees of freedom (a discrete system governed by matrix equations), making the problem solvable by a computer. The following figures show examples of FEAs performed on mechanical systems to capture deformation, stress, instability, natural frequencies, thermal distribution, and so on.





© 2023 – FEA ACADEMY

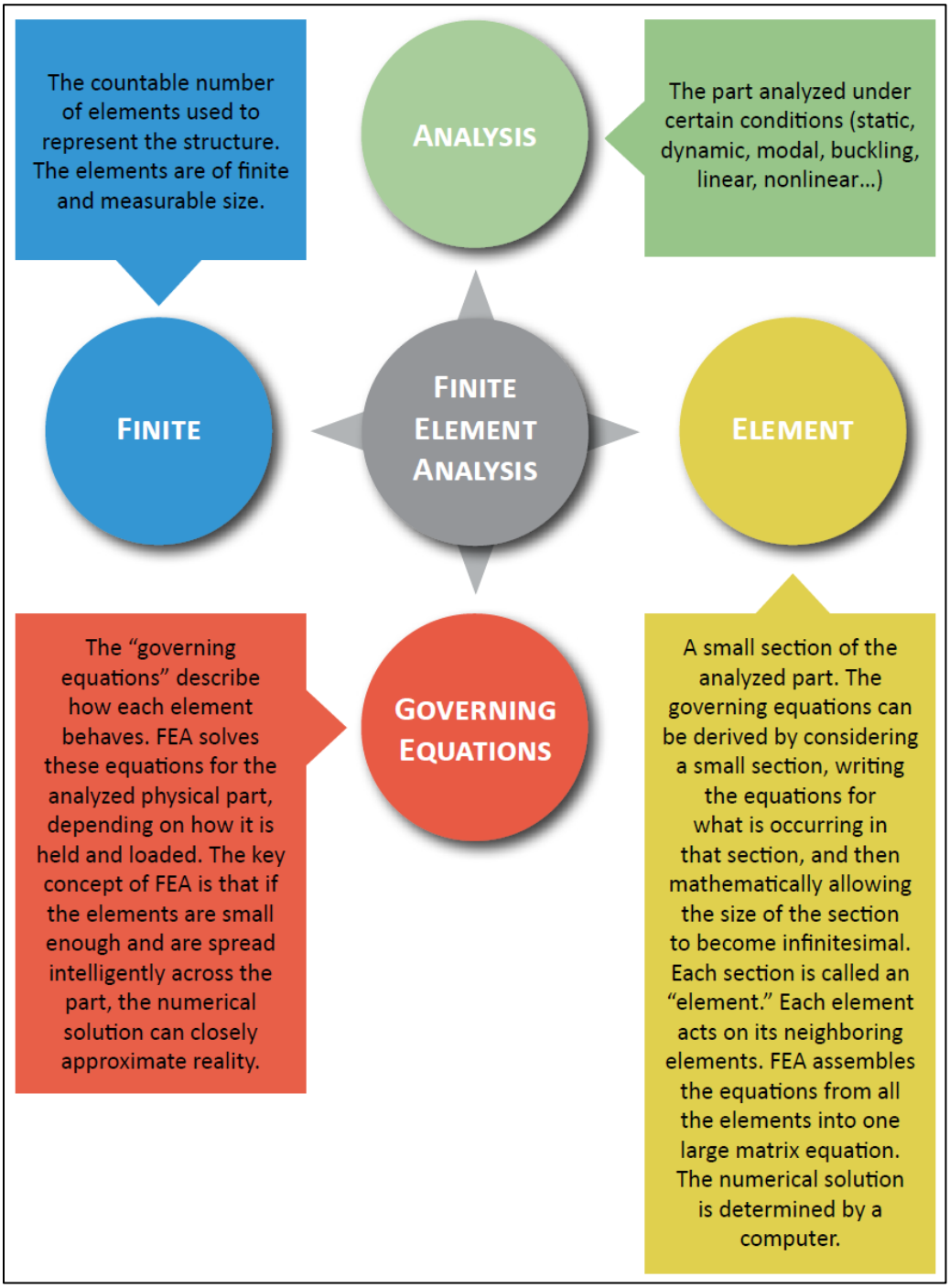
## From the Need for Analysis to the Need for Modeling






© 2023 – FEA ACADEMY

## The FEA Concept






© 2023 – FEA ACADEMY

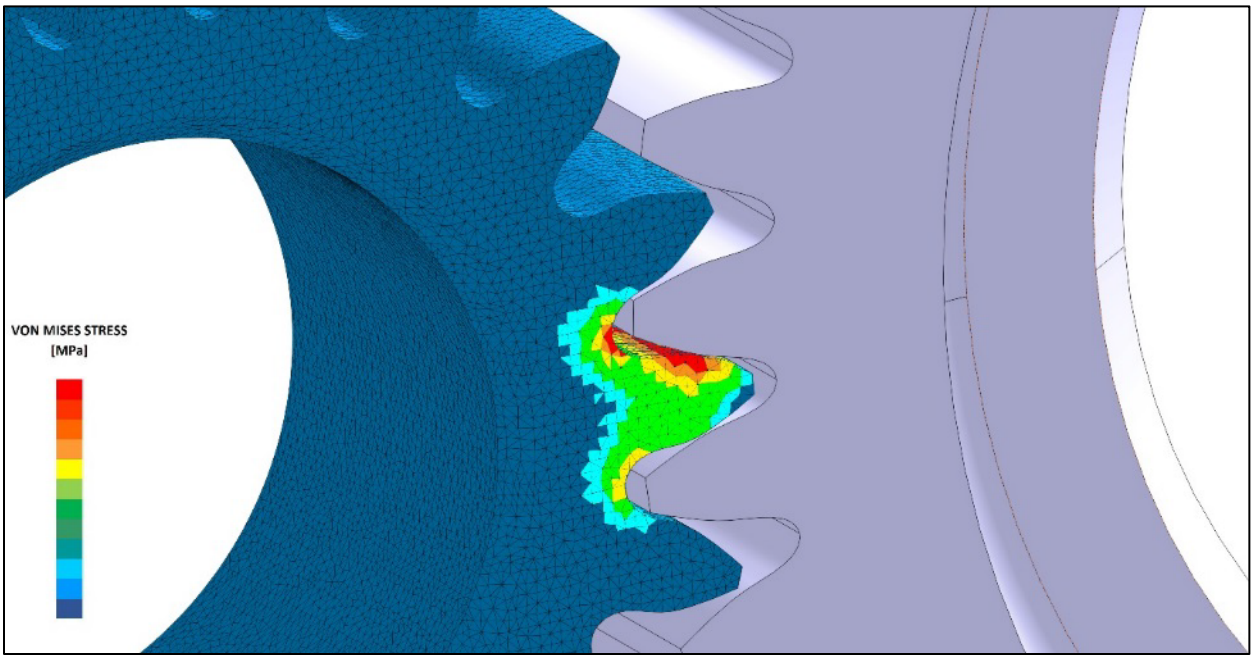
# Section 2. Understanding Mesh in FEA

This section provides a detailed and nuanced understanding of meshing in FEA, covering its types, importance, and impact on analysis. It lays the foundation for further discussions on mesh generation techniques and their practical applications in the following sections of the article.

## What is Meshing?

### Definition and Purpose

Meshing in Finite Element Analysis (FEA) is the process of breaking down a complex, continuous geometric space into smaller, manageable elements, or 'meshes'. This discretization allows for the numerical approximation of physical phenomena, translating intricate structures into a finite set of equations solvable by computers. The mesh serves as the backbone for the entire FEA process, providing a framework upon which calculations regarding stress, strain, heat transfer, and other physical attributes are conducted. It's the bridge between a theoretical model and a practical simulation, converting abstract concepts into tangible data.





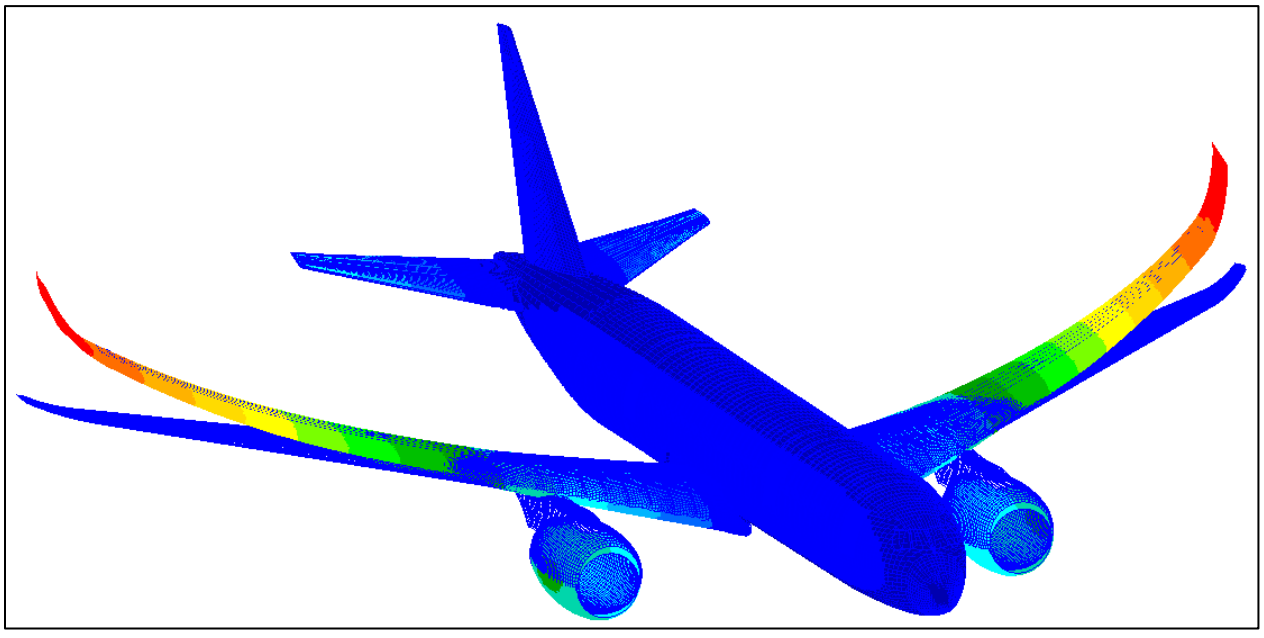

© 2023 – FEA ACADEMY

## Process Overview

The mesh generation process starts with defining the geometry of the object or space under consideration. This geometry is then partitioned into elements, with each element representing a discrete portion of the whole. The process involves decisions on the type, size, and density of the elements, significantly influenced by the nature of the physical problem being solved. Following mesh creation, boundary conditions and material properties are applied, enabling the simulation to mimic real-world scenarios accurately. The complexity of the mesh often directly correlates with the simulation's accuracy, highlighting the critical nature of this step in FEA.

## Importance of Meshing in FEA

Meshing is a critical step in FEA as it directly impacts the accuracy and reliability of the simulation results. A well-constructed mesh can accurately capture the nuances of the physical model, leading to precise simulations. Conversely, a poorly designed mesh might result in significant errors, misleading results, or even computational failure. The mesh dictates how well the simulation will replicate the real-world conditions, making it an indispensable part of the FEA process.





© 2023 – FEA ACADEMY

# Types of Elements

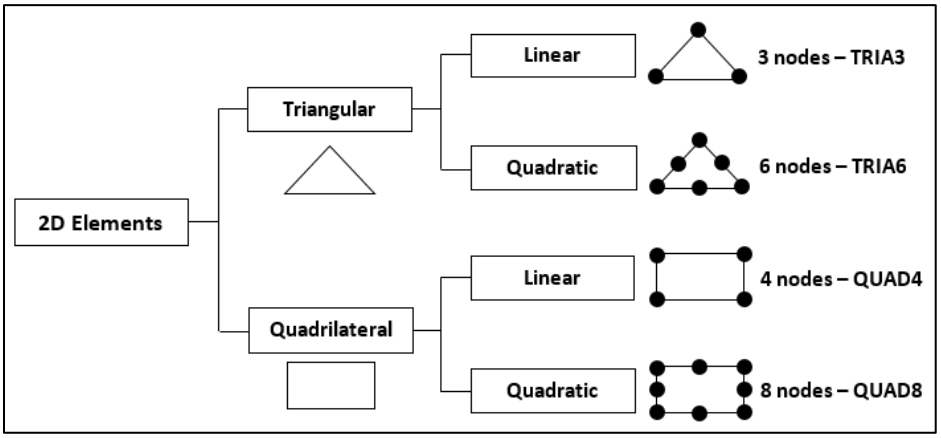
## 1D Elements (Beam)

One-dimensional (1D) elements, commonly known as beam elements, are the simplest form in FEA. They are ideal for simulating structural components that have a predominant length dimension, such as beams, trusses, and columns. Beam elements are defined by two nodes, each representing an end of the beam, and they assume linear behavior between these points. This makes them incredibly efficient for analyzing linear structures under load but limits their application in scenarios involving complex geometries or non-linear behaviors.

The primary advantage of 1D elements is their computational efficiency. Due to their simplicity, they require significantly fewer computational resources compared to higher-dimensional elements. This makes them particularly useful in large-scale structures where a detailed analysis of every component is unnecessary or impractical.

## 2D Elements (Shell)

Two-dimensional (2D) elements, or shell elements, are used to model objects with a dominant surface, like thin-walled structures, plates, and shells. These elements are defined by three or more nodes forming a flat or curved surface. Shell elements are particularly effective in analyzing stress, deformation, and vibration in structures like aircraft wings, car bodies, or pressure vessels.





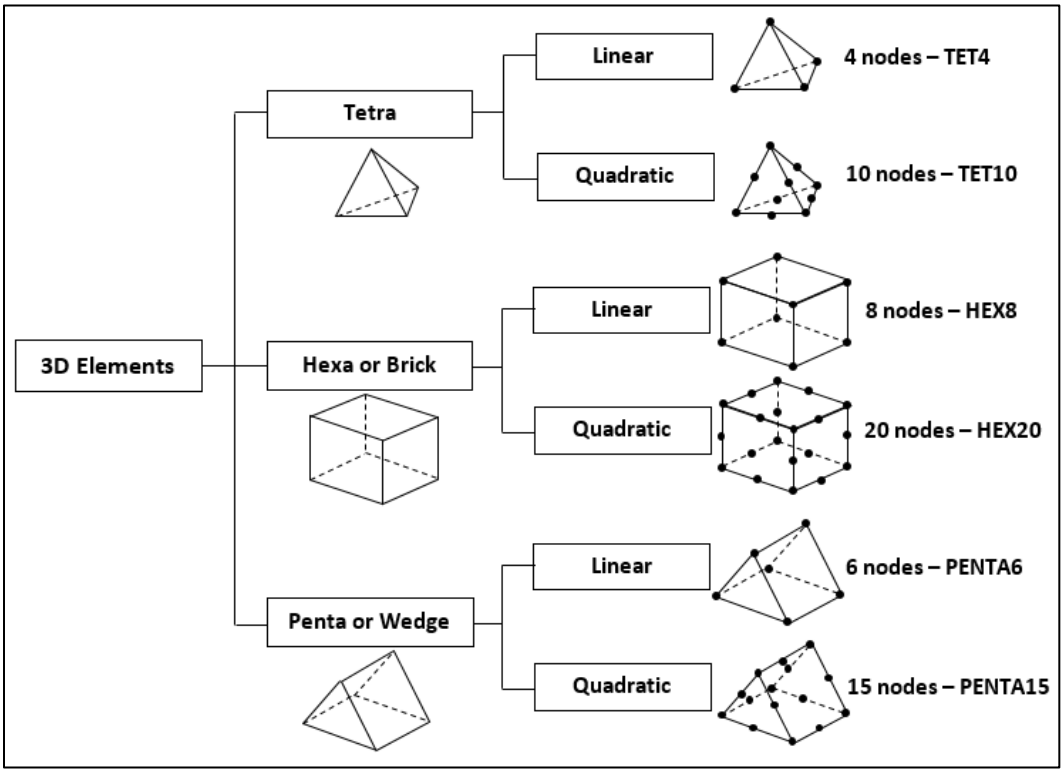

© 2023 – FEA ACADEMY

The advantage of using 2D elements lies in their ability to provide a good balance between computational efficiency and the ability to model complex behaviors. They can simulate bending, stretching, and in-plane forces with reasonable accuracy. However, a critical consideration when using shell elements is the accurate representation of thickness and curvature, which are vital for precise results.

Shell elements are versatile, but they can be challenging to use correctly. The complexity increases with the curvature of the surface and the non-uniformity of thickness. Therefore, careful consideration is required in their application to ensure accurate simulation results.

## 3D Elements (Solid)

Three-dimensional (3D) elements, or solid elements, are used for modeling objects with significant depth, width, and height. These elements are particularly useful for analyzing volumetric objects, where stresses and strains are not limited to a plane or a single axis. Solid elements are defined by four or more nodes, forming tetrahedrons, hexahedrons, or other polyhedral shapes.






© 2023 – FEA ACADEMY

Solid elements are incredibly versatile and can model complex behaviors, including non-linear material properties, large deformations, and intricate contact interactions. This versatility, however, comes at the cost of computational intensity. Solid elements require significant processing power and memory, especially when the mesh is refined for accuracy.

The use of 3D elements is essential in scenarios where the stress distribution is complex and cannot be accurately captured by 1D or 2D elements. The challenge lies in balancing the level of detail in the mesh with the available computational resources. Too coarse a mesh may overlook critical stress concentrations, while an overly fine mesh can lead to prohibitively long computation times.

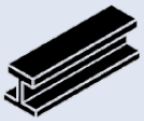
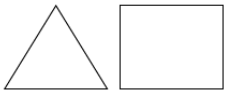
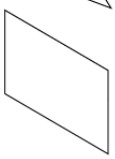
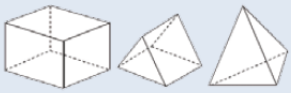
## Choosing the Right Element Type

Selecting the appropriate type of element for a given FEA task is a critical decision that depends on several factors. The geometry of the object being analyzed, the nature of the loads and constraints, and the desired accuracy of the results all play a role in this choice. For simple structures under normal loads, 1D elements may suffice, while complex structures with non-uniform loads may require 2D or even 3D elements for accurate analysis.

The choice of element type also depends on the specific requirements of the project, such as the level of detail needed in the results and the computational resources available. For instance, while 3D elements offer the most detailed analysis, they may not be feasible for large-scale projects due to the high computational demand. In such cases, a combination of different 1D and 2D element types may be employed to optimize both accuracy and efficiency.



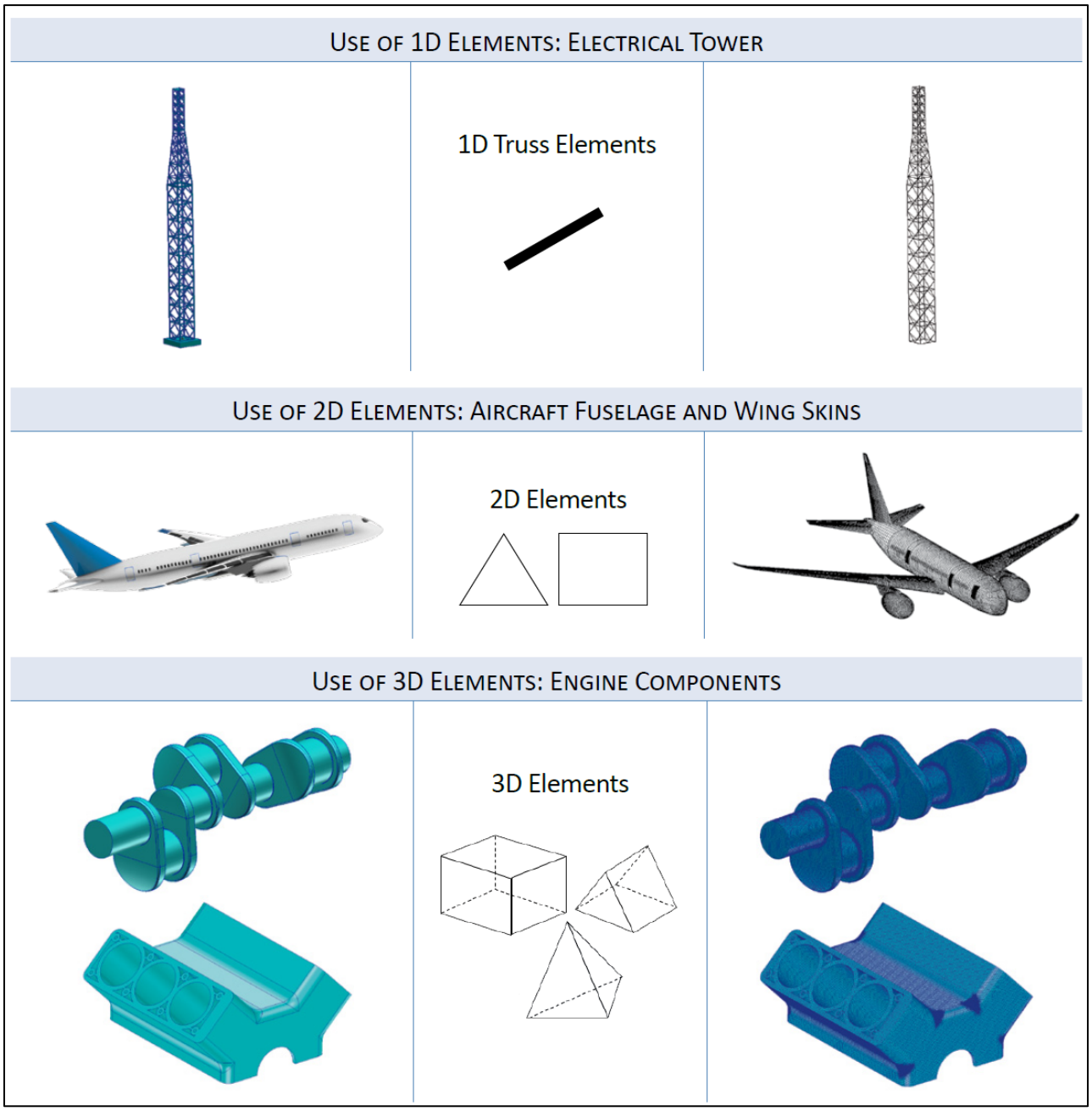
© 2023 – FEA ACADEMY

| CATEGORY | TYPE | NUMBER OF NODES | DESCRIPTION |
|---|---|---|---|
| 1D | Truss  | 2 | A long, slender element with two nodes that can be oriented in any direction in 3D space; transmits axial force only |
| | Beam  | 2 | A long, slender element with two nodes that can be oriented in any direction in 3D space; six DoFs per node allowing three translations and three rotations |
| 2D | Plain Strain of Stress  | 3 or 4 | A 2D element with only two translational DoFs per node in their plane; used in plane stress or plane strain analyses |
| | Membrane  | 3 or 4 | A 2D element that can be oriented in any direction in 3D space; transmits only in-plane loading at each node |
| | Shell  | From 3 to 8 | A 2D element that can be oriented in any direction in 3D space; transmits all translational DoFs and rotational DoFs, except the rotation around the normal of the element |
| 3D | Solid  | From 8 to 20 | A 3D solid element having various forms; transmits only translational DoFs |



© 2023 – FEA ACADEMY

## Examples of 1D, 2D and 3D Usage






© 2023 – FEA ACADEMY

## Required Dimensions for Each Type of Element

To calculate the stiffness matrix, the solver requires that all dimensions be defined. The meshing, made up of nodes and elements, is used to describe the dominant dimensions with 1D, 2D, or 3D elements selected accordingly. However, depending on the element type (1D, 2D, or 3D), the solver may require additional dimensions. The table below shows the additional dimensions required for each type of element.

| ELEMENT TYPE | USAGE | ELEMENT SHAPE | ADDITIONAL DIMENSIONS REQUIRED BY THE SOLVER | PRACTICAL USAGE |
|---|---|---|---|---|
| 1D element | Used when one dimension is very large in comparison to the other two | Line connecting two nodes | The solver considers only one of the 3 dimensions. The remaining dimensions which define the cross section, must be specified by the user as additional input data. | Shaft<br>Rod<br>Beam<br>Column<br>Bolted joint<br>Pin joint<br>Fastener |
| 2D element | Used when two of the dimensions are very large in comparison to the third | Surface connecting 3 or 4 nodes | The solver considers 2 of the 3 required dimensions. The third dimension, the thickness, must be provided by the user as additional input data. | Skin<br>Plate<br>Web<br>Parts having a width:thickness ratio > 20 |
| 3D element | Used when all three dimensions are comparable | Volume connecting 4 or 8 nodes | The solver considers all the required dimensions. | Fittings<br>Engine block |

When a geometry is borderline (somewhere between 2D and 3D geometry classification), 2D shell elements are preferred over 3D for dynamic analysis because they are less stiff and will capture the mode shapes more accurately with fewer nodes and elements.

Moreover, in nonlinear problems, the use of 3D elements may lead to too many degrees of freedom, thereby preventing the problem's resolution by the solver. This is why 1D and 2D elements are favored in nonlinear analysis, while 3D elements are only used when there is no other alternative.



© 2023 – FEA ACADEMY

# How to Choose the Right Element

## Predict Your Structure's Behavior

The critical issue when selecting elements for your FEA is knowing how the structure will behave before you begin your modeling work. You must understand the load path in your structure to be able to select the right elements. Remember that you should not employ FEA to predict the behavior of your structure. In fact, you must do the opposite and be able to predict the behavior of your structure to build a reliable and accurate FEM. If you do not have a fairly good idea of how the structure will behave, you may be misled by incorrect results based on incorrect assumptions in your element selection.

## Experiment Your Library of Elements

If you are not familiar with an element, experiment it by doing a very simple test model. This practice will give you a better understanding of an element's capability and limitations, prior to using it in a large production model. Before beginning to build your model, make sure you understand all the capabilities and limitations of the elements you use. Problems can often be solved in many different ways, but making a good choice can reduce effort, computing time, and errors in the results.

## Understanding Element Behavior

The process of meshing starts with the selection of the appropriate element types to guarantee the proper interaction with applied loads and boundary conditions. To perform this selection, a very good understanding of the behavior of each element type is essential.

To ensure you select the right element type from the library, it is important to understand which degrees of freedom are transmitted by each element because this dictates the capacity of the elements to model a given problem. The degrees of freedom associated with each element type also dictate compatibility. You should understand the library of elements associated with your FEA software before beginning any meshing activity.



© 2023 – FEA ACADEMY

## Rules for Selecting the Right Element

1. Answer the questions:
   - What is the purpose of the FEA?
   - Do you want to capture stiffness, load distribution, or stresses/strains?
   - Which solution(s) will you run with this model (static, dynamic, linear, nonlinear, normal mode, buckling)?
   - Do you know all the limitations of the library of elements of the solver you wish to run?
2. When selecting an element, make sure you understand its topology and associated assumptions and limitations (refer to the library of elements handbook for the solver you wish to use).
3. Use beam elements when shells or solids require excessive effort in meshing. This rule is related to the complexity of beam elements.
4. Model thin plate structures (thickness < 10 to 20 times the edge length) with shell elements.
5. Prefer QUAD elements over TRIA elements for the structures meshed as shells.
6. Use TRIA elements only if you do not have a choice, e.g., in transition areas. Ensure that the number of TRIA elements is negligible compared to the number of QUAD elements.
7. Mesh solid parts with 3D solid elements (HEX or TET).
8. Opt for HEX rather than TET elements in a structure meshed with solid elements. Use TET elements when the effort required to model with HEX elements is excessive.
9. Opt for TET10 rather than TET4 elements.
10. When you combine different topologies of elements such as beams, shells or solids make sure you understand all the assumptions in terms of degrees of freedom and manage the transitions.



© 2023 – FEA ACADEMY

# Mesh Quality and its Impact on Analysis

## Defining Mesh Quality

Mesh quality in FEA refers to the characteristics of the mesh that influence the accuracy and stability of the simulation results. High-quality meshes have well-shaped elements with uniform size and aspect ratios, minimal skewness, and appropriate density. The quality of a mesh is assessed using various metrics, such as element shape quality, aspect ratio, and Jacobian determinant.

A high-quality mesh effectively represents the geometry of the model, accurately captures the stress gradients, and ensures numerical stability in the analysis. Conversely, a poor-quality mesh can lead to inaccurate results, convergence issues, and increased computational time.

## Impact on Results

The quality of the mesh has a profound impact on the FEA results. A well-constructed mesh ensures that the physical phenomena being modeled are accurately represented, leading to reliable and accurate simulation outcomes. High-quality meshes capture critical stress concentrations and accurately represent the behavior of the material under various loading conditions.

On the other hand, a poor-quality mesh can result in significant errors in the simulation. These errors might manifest as unrealistic stress concentrations, incorrect deformation patterns, or convergence issues during the analysis. In extreme cases, a poor-quality mesh can lead to completely erroneous conclusions, potentially compromising the integrity of the design and safety of the structure.

Ensuring mesh quality is particularly important in simulations involving complex geometries, non-linear materials, and large deformations. In these scenarios, the mesh must be capable of capturing the intricate behaviors and interactions within the model accurately.



© 2023 – FEA ACADEMY

## Balancing Mesh Quality and Computational Efficiency

Achieving an optimal mesh is a balance between quality and computational efficiency. While a finer mesh with higher quality can yield more accurate results, it also demands more computational resources and time. Conversely, a coarser mesh reduces computational load but may compromise accuracy.

The key is to identify areas in the model where higher accuracy is critical and refine the mesh accordingly, while keeping it coarser in less critical regions. This targeted approach, known as adaptive mesh refinement, ensures that computational resources are efficiently utilized without sacrificing the accuracy of the simulation results.



© 2023 – FEA ACADEMY

# Section 3. Mesh Generation Techniques

This section covers the essential techniques in mesh generation for FEA, providing insights into their applications, advantages, and limitations. These insights help engineers and analysts make informed decisions about mesh generation, ensuring optimal results in their simulations.

## Manual vs Automated Meshing

### Manual Meshing



Manual meshing is a traditional approach where the mesh is created and refined by a user, typically an engineer or a technician. This method offers a high level of control over the mesh generation process, allowing the user to make precise adjustments to mesh density, element size, and shape according to the specific requirements of the analysis. Manual meshing is particularly advantageous when dealing with complex geometries or when specific areas of a model require focused attention.

The precision and customization offered by manual meshing can lead to higher-quality meshes in certain applications. However, this method is time-consuming and requires a significant amount of expertise and experience. The quality of the mesh heavily relies on the skill of the individual creating it, making it less consistent and potentially prone to human error. Additionally, manual meshing is not practical for large or highly complex models due to the time and labor involved.



© 2023 – FEA ACADEMY

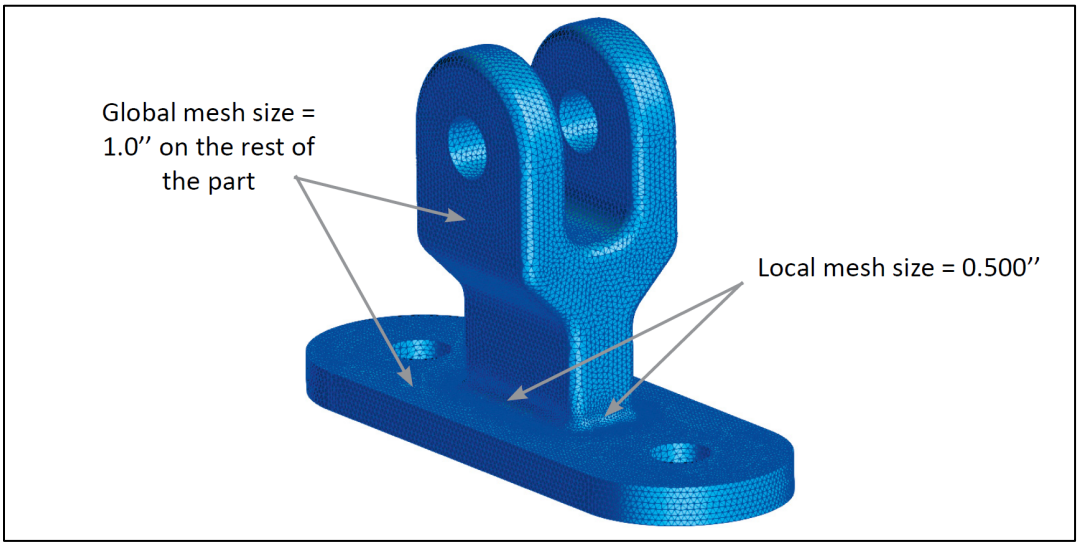
## Automated Meshing



This approach significantly reduces the time and effort required to create a mesh, making it ideal for large or routine projects. Automated meshing tools are equipped with algorithms that can handle complex geometries and are capable of automatically refining the mesh in areas where higher accuracy is needed.

While automated meshing is less labor-intensive and provides faster results, it may not always produce the optimal mesh for every scenario. The quality of the mesh generated by automated methods can vary depending on the sophistication of the software and the parameters set by the user. It's crucial to have a good understanding of the software's capabilities and limitations to effectively use automated meshing.

In practice, a hybrid approach is often used, where the initial mesh is generated automatically, and manual refinements are made as needed. This approach combines the efficiency of automated meshing with the precision of manual adjustments, leading to a balanced and effective meshing process.






© 2023 – FEA ACADEMY

# 2D Shell Meshing

## Overview of 2D Shell Meshing

2D shell meshing is a specialized technique used in Finite Element Analysis for modeling thin-walled structures (any parts having a width:thickness ratio > 20). These structures include components like vehicle bodies, aircraft fuselage, and other objects where one dimension (thickness) is significantly smaller than the other two. Shell elements in 2D meshing are defined by their surface geometry, with each element typically having three or four nodes and possessing both bending and in-plane stiffness.

Shell meshing is particularly effective in applications where structural behavior is dominated by bending and in-plane forces. This technique offers a balance between computational efficiency and the ability to capture complex behaviors of thin-walled structures. It allows for the analysis of large structures, like entire aircraft wings or car bodies, with reasonable computational resources.

## Advantages of 2D Shell Meshing

One of the primary advantages of 2D shell meshing is its ability to efficiently model large structures with high accuracy. By focusing on the surface geometry, shell elements can accurately capture the structural response to loads without the need for a volumetric (3D) mesh, which significantly reduces the number of elements and computational effort. Additionally, shell meshing can accommodate a variety of material models, including orthotropic and composite materials, making it ideal for a wide range of engineering applications.

## Challenges and Considerations

However, shell meshing also presents certain challenges. Accurately defining the thickness and material properties of the shell is crucial, as these directly impact the analysis results. The transition regions, where the shell meets a solid or another shell with different properties or thickness, require careful meshing and consideration. It's also essential to correctly apply boundary conditions and loading to shell elements to avoid inaccuracies.

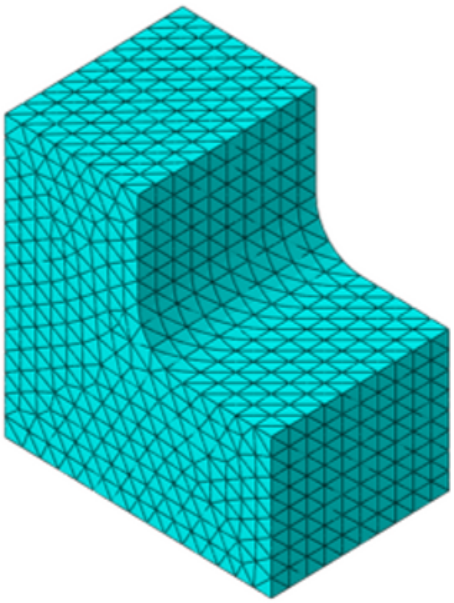


© 2023 – FEA ACADEMY

In summary, 2D shell meshing is a powerful tool in FEA, offering a practical and efficient method for analyzing thin-walled structures. While it simplifies the modeling of complex geometries, attention to detail in defining properties and boundary conditions is key to obtaining accurate results.

## Tetrahedral vs Hexahedral 3D Meshing

### Tetrahedral Meshing



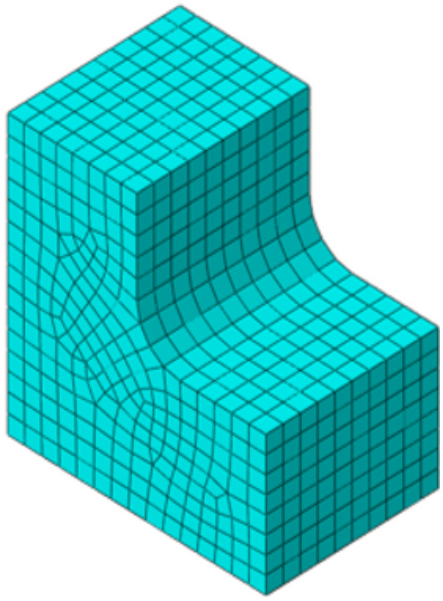
Tetrahedral meshing involves dividing the domain into tetrahedrons – three-dimensional elements with four triangular faces. This type of meshing is highly versatile and can easily conform to complex geometries, making it a popular choice in many FEA applications. Tetrahedral elements are particularly useful in scenarios where the geometry is too intricate for simpler element shapes.

The main advantage of tetrahedral meshing is its flexibility and ease of generation, especially in automated meshing environments. However, tetrahedral elements can be less efficient and less accurate than hexahedral elements, especially for bending and twisting loads. They often require a finer mesh to achieve the same level of accuracy, which can increase computational requirements.



© 2023 – FEA ACADEMY

## Hexahedral Meshing



Hexahedral meshing, on the other hand, uses six-faced elements (hexahedrons) and is often preferred for its superior accuracy and efficiency in many applications. Hexahedral elements are particularly effective in modeling bending and twisting phenomena and generally provide more accurate stress and strain results with fewer elements compared to tetrahedrons.

The challenge with hexahedral meshing lies in its application to complex geometries. Creating a high-quality hexahedral mesh manually can be time-consuming and requires significant expertise. Automated hexahedral meshing is also more challenging and less robust than tetrahedral meshing due to the geometric constraints of hexahedral elements.

In practice, the choice between tetrahedral and hexahedral meshing depends on the specific requirements of the analysis, including the geometry of the model, the type of loading, and the desired level of accuracy.



© 2023 – FEA ACADEMY

# Advanced Techniques: Sweeping, Paving, and Mapped Meshing

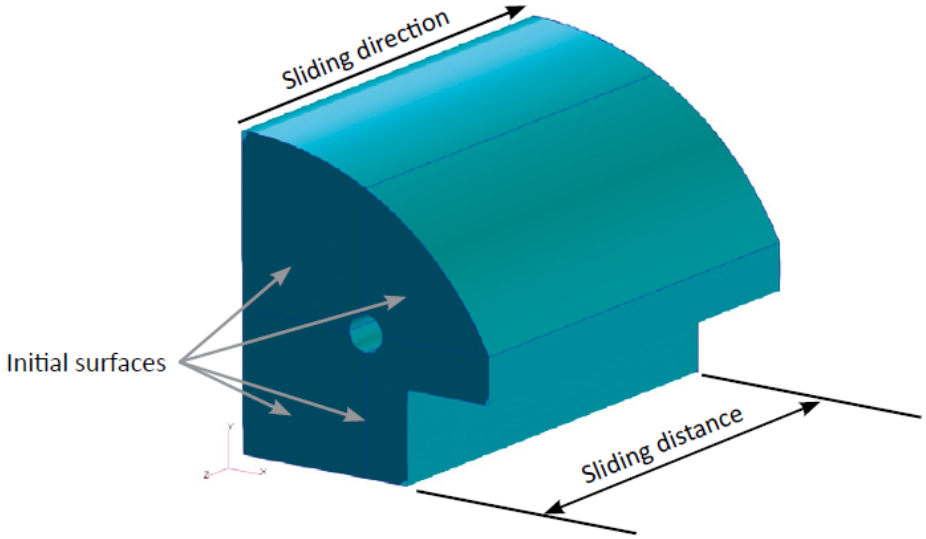
## Sweeping

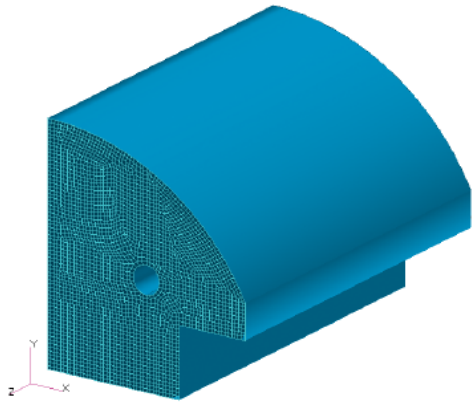
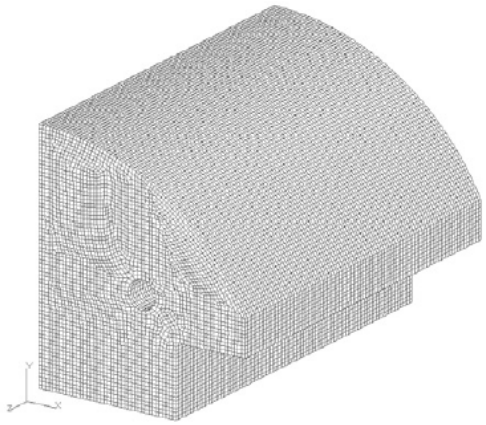
Sweeping is a mesh generation technique used for objects with a consistent cross-section along a particular direction. In this method, a 2D mesh is created on the cross-sectional area, which is then 'swept' along a path to create a 3D mesh. This technique is highly efficient for elongated objects like beams, channels, and pipes, as it produces high-quality meshes with fewer elements.

The sweeping technique is particularly advantageous for its ability to generate well-structured, high-quality meshes quickly. However, its applicability is limited to geometries that have a uniform cross-section along one dimension.



© 2023 – FEA ACADEMY




| | |
|---|---|
| Step 1: generate a 2D shell mesh on the initial surface of the solid. At this stage, the mesh density of the final HEX mesh will be influenced by the 2D mesh. |  |
| Step 2: sweep the 2D mesh by specifying a vector for the direction and length of the extrusion and define the number of necessary HEX elements along the vector, or their dimensions. |  |



© 2023 – FEA ACADEMY

## Paving

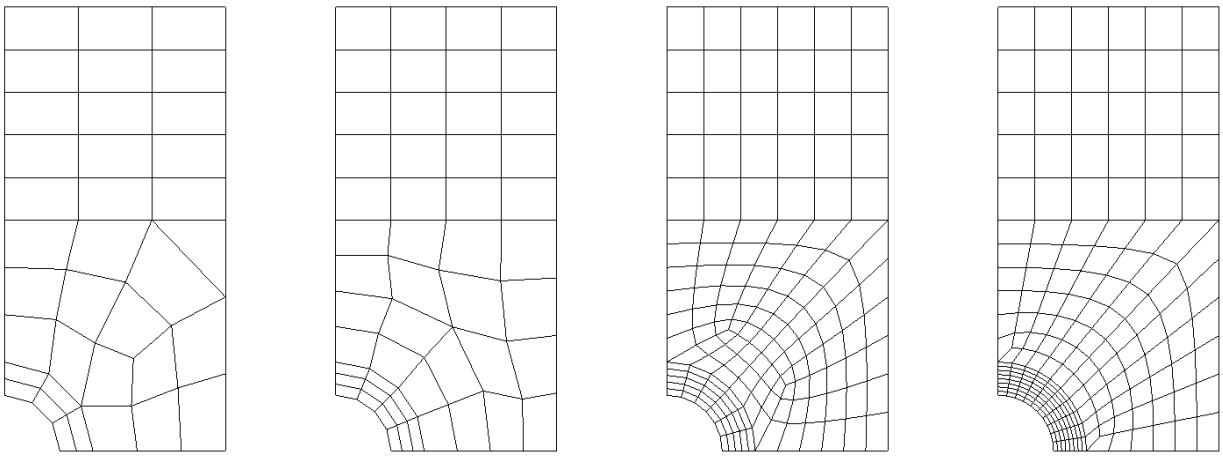
Paving is a technique primarily used for 2D meshing, especially in areas with irregular boundaries. It involves laying out elements in layers, starting from the boundaries and working towards the interior, somewhat akin to paving stones in a path. This technique is effective in creating a well-structured mesh that conforms closely to the boundaries of the model.

Paving is beneficial for its ability to handle complex boundary shapes and to create a uniform mesh. However, like sweeping, its use is somewhat limited by the geometry of the model.

## Mapped Meshing

Mapped meshing is a technique used for objects with regular, geometrically simple shapes, such as rectangles or cylinders. In mapped meshing, the domain is divided into regular sub-domains, and a structured mesh is created within each sub-domain. This method allows for a high degree of control over element size and shape, resulting in a uniform and efficient mesh.

Mapped meshing is particularly effective for models with simple geometries where a high-quality, structured mesh is desired. However, its application is limited to objects that can be easily subdivided into regular shapes.





© 2023 – FEA ACADEMY

## Why Mesh in 2D Instead of 3D?

I have often seen thin-walled structures modeled with 3D solid elements simply because the FEA analyst wished to decrease the extraction time required for the mid-planes or did not know how to extract them. This is an incorrect approach that produces inaccurate results. The main problem with using 3D solid elements to model thin-walled structures is shear locking. Shear locking is an over-stiff behavior that results when energy is expended on shearing the element, rather than bending. This locking phenomenon is even more pronounced when the solid elements look like shells (one dimension that is much smaller than the other two). This problem may be circumvented by placing three or four elements across the thickness to capture all bending and stiffness effects and generate an accurate solution. However, this approach produces a tremendous number of degrees of freedom.

Therefore, for thin-walled structures such as skins, pressure vessels, ribs, bulkheads, webs, and so on, 2D shell elements are the only option, of which the preferred choice is the linear quadrilateral element.

Moreover, if a high-quality mesh is required, a shell mesh is easier to create than a solid one. From an analytical point of view, the shell model also offers the great advantage of using fewer resources (less RAM, disk space, and CPU time), especially for nonlinear analysis and larger models. This is because, rather than actually modeling the thickness with additional nodes and elements, shell models capture wall thickness as a mathematical value, resulting in fewer equations to solve.

Another benefit of 2D shell models over 3D solid models is the ease with which you can change the thickness in your simulations, since the thickness is a property associated with the shell element. With a 3D model, you will have to move nodes or even re-mesh certain parts. Therefore, the shell model is a better choice for conducting sensitivity studies.

Finally, post-processing with shell elements is easier and faster than with solid elements, which is not a minor detail, when you need to post-process numerous load cases.



© 2023 – FEA ACADEMY

# Section 4. Mesh Density and Element Size

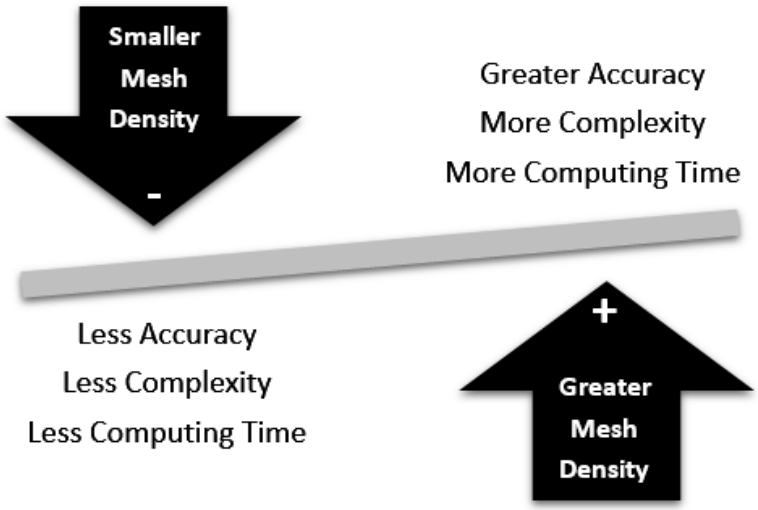
This section comprehensively covers the critical aspects of mesh density and element size in FEA, providing insights into their determination, importance, and the balance between accuracy and computational demands. These principles are fundamental to creating effective and efficient FEA models.

## Determining Appropriate Mesh Density

### Understanding Mesh Density

Mesh density in Finite Element Analysis refers to the number of elements within a given area or volume of the model. It's a crucial factor that directly influences the accuracy and computational intensity of the analysis. High mesh density means more elements are used to represent the model, allowing for a more detailed and accurate representation of the physical phenomena. However, this also leads to increased computational resource requirements and longer processing times.

Determining the appropriate mesh density depends on several factors, including the complexity of the geometry, the type of loading, the expected stress gradients, and the criticality of the simulation's accuracy. Areas with high stress gradients or significant changes in geometry typically require higher mesh density to capture the behavior accurately.






© 2023 – FEA ACADEMY

## Strategies for Determining Mesh Density

There are several strategies to determine the optimal mesh density for an FEA model. One common approach is to start with a coarse mesh and gradually refine it in areas where higher accuracy is required. This can be guided by stress analysis results, where higher stress concentrations indicate the need for finer mesh.

Another approach is to use adaptive meshing techniques, where the mesh density is automatically adjusted based on the stress gradients or error estimates. This method is particularly effective in complex analyses, as it dynamically refines the mesh in regions where it's most needed.

In addition to stress analysis, considering the type of material behavior, such as plastic deformation or large strains, is also crucial in determining mesh density. These material behaviors often require finer meshes to accurately capture the nonlinear response of the material.

# Importance of Element Size and Aspect Ratio

## Element Size Considerations

The size of the elements in a mesh is a key factor in the quality of the FEA simulation. Smaller elements generally provide more detailed results but at the cost of increased computational effort. The choice of element size should be based on the level of detail required in the simulation results and the geometric complexity of the model.

In areas with complex geometry or high stress concentrations, smaller elements are necessary to accurately capture the behavior. However, in regions where stresses are relatively uniform or less critical, larger elements can be used to reduce computational demands.



© 2023 – FEA ACADEMY

## Aspect Ratio and its Impact

The aspect ratio of an element, defined as the ratio of its longest to shortest dimension, is another critical factor in mesh quality. Elements with high aspect ratios (elongated elements) can lead to inaccurate results, especially in stress calculations. Ideally, elements should be as close to equilateral as possible, particularly in areas of high stress or complex material behavior.

Maintaining a low and uniform aspect ratio throughout the mesh ensures that the elements can accurately represent the physical phenomena, providing reliable results. However, in some scenarios, such as thin structures or areas with sharp geometrical features, high aspect ratio elements might be unavoidable. In these cases, special care should be taken in interpreting the results, and additional validation may be necessary.

# Balancing Accuracy and Computational Efficiency

## Achieving an Optimal Balance

Balancing the accuracy of the simulation with the computational efficiency is a key challenge in FEA. A highly detailed mesh (high density and small element size) will yield accurate results but requires significant computational resources and time. Conversely, a coarser mesh reduces the computational load but may compromise the accuracy and reliability of the results.

To achieve an optimal balance, it's essential to understand the critical aspects of the model and the analysis objectives. For instance, in preliminary design stages, a coarser mesh may be sufficient to get an overall understanding of the structural behavior. However, for detailed design verification or failure analysis, a finer mesh is necessary to accurately capture critical stress concentrations and complex behaviors.



© 2023 – FEA ACADEMY

## Techniques for Balancing

Several techniques can be employed to balance accuracy and computational efficiency. One approach is to use multi-scale analysis, where different levels of mesh detail are used for different parts of the model, concentrating finer meshes in critical areas. Another technique is to use higher-order elements, which can capture more complex behaviors with fewer elements, thus reducing the computational burden.

Additionally, modern FEA software often includes tools for optimizing the mesh based on the desired balance between accuracy and efficiency. These tools can automatically adjust the mesh density and element size based on predefined criteria, ensuring that the mesh is as efficient as possible without sacrificing the necessary accuracy.

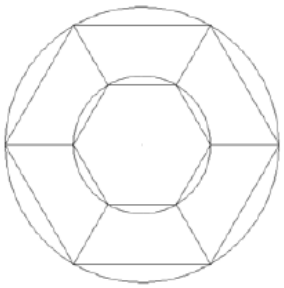
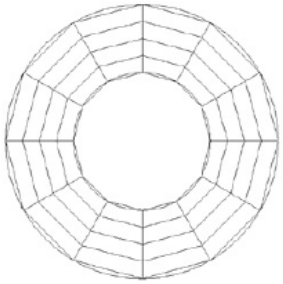
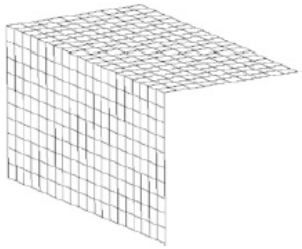
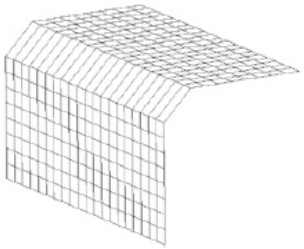
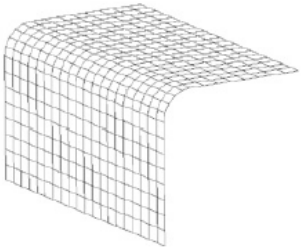
In summary, determining the appropriate mesh density, understanding the importance of element size and aspect ratio, and balancing accuracy with computational efficiency are critical aspects of mesh generation in FEA. These factors significantly influence the quality of the simulation results and the overall effectiveness of the analysis. A well-designed mesh, tailored to the specific requirements of the model and analysis, is essential for obtaining reliable and accurate simulation outcomes.



© 2023 – FEA ACADEMY

## Rules for Modeling Holes and Fillets

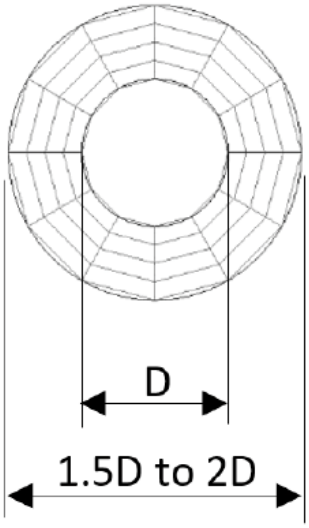
Holes and fillets are common geometry types that sometimes require more precise meshing to capture local effects. The table below summarizes the rules for the general and critical regions. The rules for critical regions apply if peak stresses or strains are extracted, while the rules for general regions apply to regions where stresses or strains are not extracted but where we nevertheless want to model the proper stiffness.

<table>
<tr><th></th><th>GENERAL REGION</th><th>CRITICAL REGION</th></tr>
<tr><td>HOLES</td><td>6 elements<br>  </td><td>At least 12 elements<br>  </td></tr>
<tr><td></td><td colspan="2">Note that to better represent the hole geometry and obtain smooth mesh flow lines, an even number of elements should be used around a hole, for example, 6, 8 10, 12, 16... elements, not 5, 7, 9, 13, and so on.</td></tr>
<tr><td>FILLETS</td><td>No element for small fillets<br>  </td><td>One element for large fillets<br>  </td><td>Between 3 and 5 elements in the radius<br>  </td></tr>
</table>



© 2023 – FEA ACADEMY

The refined mesh around a hole should be 1.5 to 2 times the hole diameter, and a minimum of two layers of fine mesh around the hole is required.






© 2023 – FEA ACADEMY

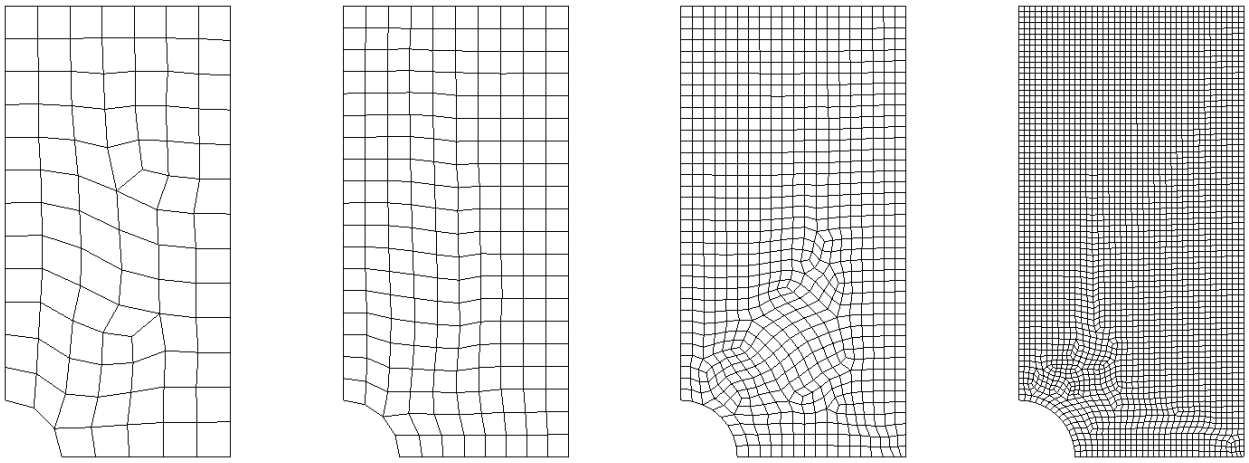
# Section 5. Mesh Refinement Techniques

This section thoroughly discusses mesh refinement techniques in FEA, highlighting their importance, applications, and practical considerations. It provides a foundational understanding of how different refinement strategies can be effectively employed to enhance the accuracy and efficiency of FEA simulations.

## Local vs Global Mesh Refinement

### Global Mesh Refinement

Global mesh refinement involves uniformly increasing the density of the mesh across the entire model. This method is straightforward and can be effective for small to medium-sized models or when a uniform level of detail is required throughout the analysis. Global refinement enhances the overall resolution of the simulation, leading to more accurate results. However, it also significantly increases the number of elements, which can exponentially raise computational costs and time, making it less practical for larger or more complex models.



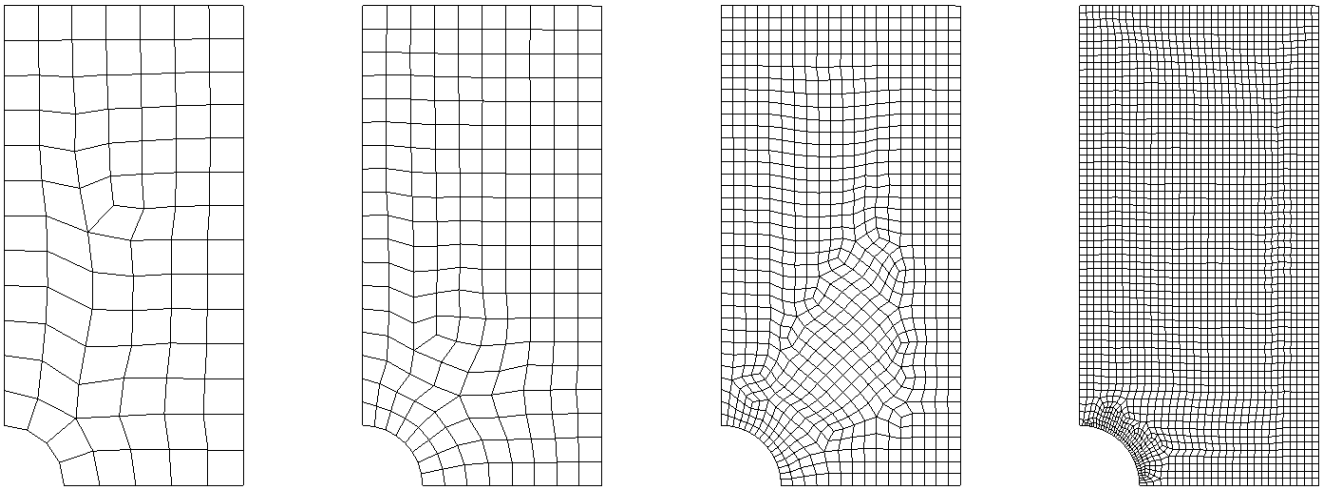


© 2023 – FEA ACADEMY

## Local Mesh Refinement

Local mesh refinement, in contrast, focuses on increasing mesh density in specific areas of the model where higher accuracy is required. This technique is particularly useful in regions with high stress concentrations, complex geometries, or near boundary conditions and load applications. Local refinement allows for detailed analysis of critical areas without the unnecessary computational burden of refining the entire mesh. This targeted approach is more efficient and practical, especially for large-scale or complex models.

The challenge with local refinement lies in identifying the regions that require it. Typically, areas with high stress gradients, potential for material non-linearities, or geometric complexities are candidates for local refinement. Effective use of this technique requires a good understanding of the model's behavior and the specific requirements of the analysis.





© 2023 – FEA ACADEMY

# Adaptive Mesh Refinement (AMR)

## Concept and Advantages

Adaptive Mesh Refinement (AMR) is an advanced technique that dynamically adjusts the mesh during the analysis based on specific criteria, such as stress gradients, strain energy, or error estimates. AMR begins with a coarse mesh and refines it iteratively in areas where the analysis indicates higher detail is needed. This approach ensures that the mesh density is optimized throughout the simulation, leading to more accurate results without unnecessarily large computational costs.

AMR is particularly effective in nonlinear analysis, transient simulations, and problems with evolving geometries, such as crack propagation or large deformations. It automates the refinement process, reducing the need for user intervention and expertise in determining where refinement is needed.

## Implementation Challenges

While AMR offers significant advantages, its implementation can be challenging. It requires sophisticated software algorithms capable of assessing the analysis results in real-time and making intelligent decisions about where and how to refine the mesh. The criteria for refinement must be carefully chosen to ensure that the mesh adapts effectively to the evolving requirements of the simulation.

AMR can also lead to non-uniform meshes with a wide range of element sizes, which might introduce additional complexities in the analysis, especially in terms of element compatibility and numerical stability. Therefore, careful consideration and validation are necessary when using AMR to ensure the reliability of the results.



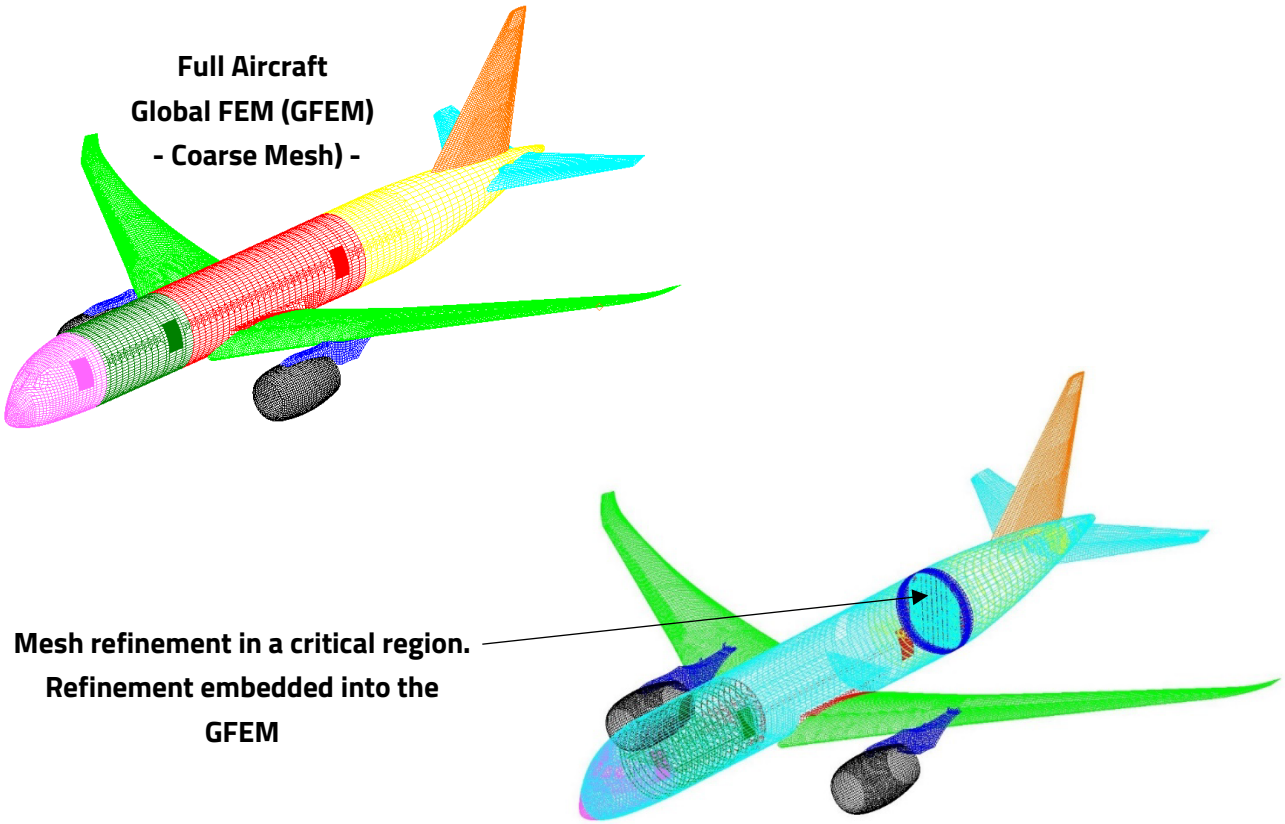
© 2023 – FEA ACADEMY

# Practical Examples of Refinement Strategies

## Case Studies and Applications

Practical examples and case studies of mesh refinement strategies offer valuable insights into their real-world applications. For instance, in automotive crash simulations, local refinement is often used around areas of expected high deformation, such as crumple zones, while keeping the rest of the mesh coarser. This approach provides detailed information about the deformation and stress distribution in critical areas while keeping the overall computational cost manageable.

In aerospace applications, global refinement might be used in preliminary design stages to get a uniform understanding of the stress distribution across the entire structure. As the design progresses, local refinement can be employed in areas like joints, connections, and areas with high stress concentration to provide more detailed analysis for design optimization.






© 2023 – FEA ACADEMY

## Best Practices and Guidelines

The choice of refinement strategy should be guided by the specific requirements of the analysis and the limitations of computational resources. It's essential to start with a clear understanding of the objectives of the simulation and the critical areas of the model. Regularly reviewing the analysis results can provide valuable feedback on the effectiveness of the mesh and guide further refinements.

In practice, a combination of global and local refinement is often used, starting with a globally refined mesh for an initial understanding, followed by targeted local refinement in critical areas. The use of AMR can further enhance this process by automating the refinement based on the evolving needs of the simulation.

## Convergence Study Methodology

By definition, a converged mesh, from a numerical accuracy standpoint, is one that produces no significant differences in the result when mesh refinement is introduced. Mesh convergence is a concern when producing a model to capture peak stress or strain. If test results such as strain gauge records exist for the part you wish to model, it is easy to determine the mesh density needed to capture the correct stress. Unfortunately, you will usually not have such records prior to beginning the meshing process. Therefore, a convergence study must be conducted for the regions of peak stress to ensure that the mesh size is sufficiently fine to correctly capture the phenomena of interest and critical stress.

A powerful method used to determine the converged mesh is the convergence curve. First, you must select a critical result parameter. For the mechanical structural engineer, it is usually stress or strain. You should conduct a convergence study for each region of the model for which you wish to extract the peak stress or strain.

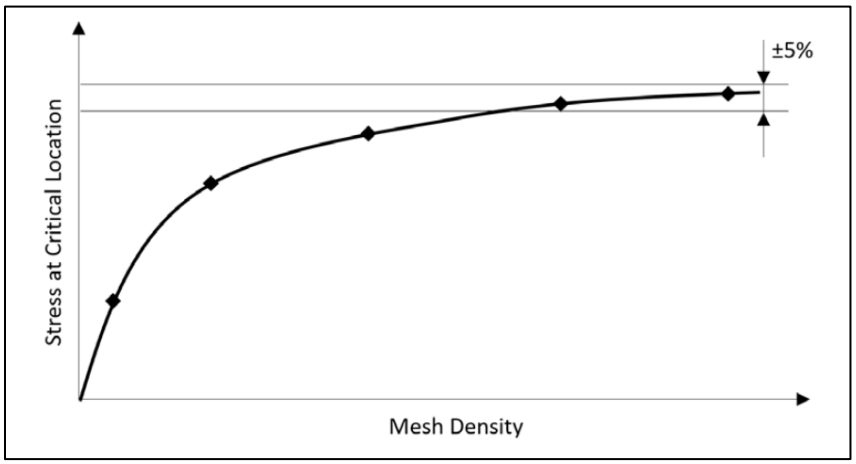
Increasing the number of nodes improves results accuracy, but it also increases the solution time and cost. The usual practice is to increase the number of elements only in regions of high stress, instead of reducing the global element size of the entire model, and to continue until the difference between the two consecutive results is less than 5%. A convergence study is the process of solving a model with successively finer meshes and comparing the results of the different meshes. This comparison is usually made by



© 2023 – FEA ACADEMY

inspecting the fields at one or more points in the model. By comparing the results obtained during mesh refinement, it is possible to judge the convergence of the solution, with respect to mesh refinement. After a minimum of three successive solutions, asymptotic behavior may begin to emerge, and variations in the results among the meshes will become smaller. If these variations are small enough, the mesh can be considered converged. This is always a judgment call on the part of the FEA analyst, who knows the uncertainties in the model inputs and the acceptable uncertainty in the results.

The figure below shows a typical convergence curve obtained during a convergence study. The last two points show that the model can be considered to be converged, so the mesh corresponding to the penultimate point should be used for the stress-strain extraction.




At least three runs are required to plot a curve, which can then be used to indicate that convergence has been achieved. However, if two runs of different mesh densities give the same result, convergence can be considered to have already been achieved, and no further mesh refinement is required.

Another check to perform during the convergence study concerns the gradient in neighboring elements: the correct gradient has not been properly captured if the neighboring elements show significant variations in stress, and additional refinement is then required.

Currently, some codes are able to compute convergence based on criteria defined by the FEA analyst. These codes are able to automatically re-mesh non-converged regions of an FEM. However, such



© 2023 – FEA ACADEMY

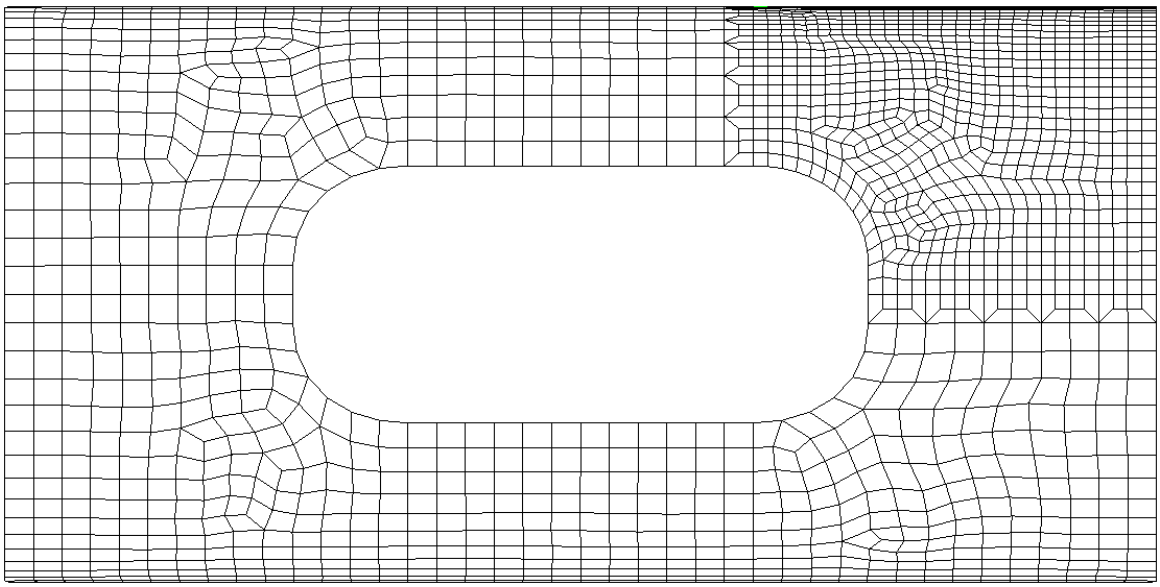
automatic convergence studies are dependent on the load cases and will produce different meshes for different ones. If the convergence study is performed manually by refining the critical regions step by step, computation of the asymptotic value is time-consuming.

## Over What Distance is the Mesh Refined?

In theory, for each successive level of mesh refinement in the convergence study, all elements in the model should be split in all directions. However, the Saint-Venant's principle implies that local stresses in one region of a structure do not affect stress elsewhere (the behavior depends on only the tensor of the internal forces, at this point).

Therefore, in practical terms, we can conduct the convergence study only in the region of interest and keep the rest of the model unconverged. At the same time, there should be transition regions to connect the refined mesh to the coarse mesh. From a practical point of view, mesh refinement is required as long as the field gradient is significant. Once it is not, the mesh transition can start.

The figure below illustrates this principle. The peak stress must be captured in the corner of the cutout, so only this region of the model is refined.





© 2023 – FEA ACADEMY

# Section 6. Common Pitfalls in Mesh Generation

This section provides a comprehensive discussion of common pitfalls in mesh generation for FEA, highlighting the challenges and providing guidance on how to avoid them. Understanding these pitfalls is crucial for ensuring accurate and efficient finite element models, and avoiding common errors that can lead to unreliable results or inefficient analyses.

## Over-refinement and Under-refinement Issues

### The Impact of Over-refinement

Over-refinement occurs when the mesh is too dense for the requirements of the analysis, resulting in an excessive number of elements. While this might seem advantageous for accuracy, it leads to unnecessarily long computation times and can exhaust computational resources. In extreme cases, over-refinement can cause numerical instabilities due to issues like ill-conditioning in the stiffness matrix and rounding errors.

The key to avoiding over-refinement is understanding the level of detail required for the analysis. Over-refinement is often unnecessary in regions of the model where stress gradients are low or where high precision is not critical. A balanced approach, focusing refinement efforts where they are most needed, is more effective and efficient.

### Challenges of Under-refinement

Under-refinement, on the other hand, is characterized by a mesh that is too coarse, with too few elements to accurately capture the behavior of the model. This can lead to inaccurate results, such as missed stress concentrations, incorrect deformation patterns, or even non-convergence of the solution. Under-refinement is particularly risky in areas with complex geometries, near boundary conditions, or in regions where material nonlinearities are expected.

To prevent under-refinement, it is essential to start with a mesh that is sufficiently detailed for the given analysis and refine it based on preliminary results or known areas of concern. Adaptive mesh refinement techniques can be particularly useful in automatically identifying and refining under-refined areas.



© 2023 – FEA ACADEMY

## Inappropriate Element Choice

### Selecting the Wrong Element Type

One of the common pitfalls in mesh generation is the inappropriate selection of element types. Different elements (1D, 2D, 3D) have specific applications and limitations. For instance, using beam elements (1D) for a problem that requires capturing complex three-dimensional stress patterns can lead to erroneous results. Similarly, employing solid elements (3D) for thin-walled structures where shell elements (2D) would be more appropriate not only increases computational effort but also potentially reduces accuracy.

The choice of element type should be based on the geometry of the part, the nature of the loads and constraints, and the type of analysis being performed. It's also important to consider the material properties and the expected mode of failure or deformation. Utilizing the correct element type is crucial for an accurate and efficient analysis.

### Avoiding Element Mismatch and Inconsistencies

In addition to choosing the appropriate type of element, ensuring consistency in the element formulation across the model is vital. Mismatched elements, where different types or formulations are used in adjacent regions without proper transition, can lead to inaccuracies and convergence issues. This is especially true in areas where different types of elements meet, such as where solid elements interface with shell elements.

## Ignoring Geometric Nonlinearities and Complexities

### Overlooking Geometric Nonlinearities

Geometric nonlinearities occur when deformations are large enough to cause significant changes in the geometry of the structure during analysis. Ignoring these nonlinearities in mesh generation can result in inaccurate modeling of the behavior under load, particularly in structures prone to large deflections or rotations.



© 2023 – FEA ACADEMY

When dealing with geometric nonlinearities, the mesh must be able to accommodate the large deformations without becoming overly distorted. This often requires finer meshing in regions expected to undergo large deformations, as well as the use of elements that are capable of capturing nonlinear geometric effects.

## Addressing Complex Geometries

Complex geometries, such as sharp corners, thin ribs, or small features, pose significant challenges in mesh generation. These features can create stress concentrations that are easily missed with a coarse mesh. At the same time, overly refining these areas can lead to over-refinement problems.

The key to successfully meshing complex geometries is to use a combination of element types and refinement techniques that can accurately represent the geometry without excessive computational cost. This might involve local refinement around complex features, the use of higher-order elements that can better capture curvature and gradients, or employing techniques like mesh smoothing to avoid irregular element shapes.



© 2023 – FEA ACADEMY

# Section 7. Mesh Validation and Error Checking

This section provides a comprehensive overview of mesh validation and error checking in FEA, covering essential techniques and real-world applications. Understanding and implementing these practices are crucial for achieving accurate and reliable simulation results, ensuring the success of engineering projects across various industries.

## Techniques for Mesh Quality Assessment

### Overview of Mesh Quality Metrics

Mesh quality assessment is crucial in FEA to ensure that the mesh accurately represents the physical model and is suitable for analysis. Various metrics are used to evaluate mesh quality, including aspect ratio, skewness, Jacobian, and warpage. A high-quality mesh typically has low aspect ratios, minimal skewness, positive Jacobian values, and limited warpage, ensuring that elements are well-shaped and capable of accurately capturing the physical behavior of the model.

Various quality checks are available in pre-processing software to help you decide if your mesh meets the criteria for analysis. Typical quality checks for 2D and 3D elements include the following (the names may differ somewhat among software):

- Aspect ratio
- Taper
- Skew angle
- Warp angle
- Interior angle
- Face Skew
- Jacobian



© 2023 – FEA ACADEMY

### Tools for Mesh Quality Evaluation

Most FEA software packages include tools for mesh quality assessment, providing visual and quantitative ways to evaluate the mesh. These tools allow engineers to identify elements that do not meet specific quality criteria and make necessary adjustments. Common practices include highlighting elements with high aspect ratios or skewness and assessing the distribution of element sizes and shapes across the model.

Using these tools, engineers can refine the mesh in targeted areas, ensuring that the overall mesh quality is within acceptable limits. This process is iterative, often requiring several rounds of refinement and reassessment to achieve an optimal mesh.

## Error Estimation and Convergence Testing

### Understanding Error Estimation

Error estimation in FEA is a method to quantify the uncertainty or potential error in simulation results due to the discretization of the mesh. This estimation helps in determining how close the FEA results are to the 'true' solution. Common techniques for error estimation include energy norm methods, which evaluate the difference in strain energy between the meshed model and an ideal solution.

### Importance of Convergence Testing

Convergence testing is a critical part of mesh validation. It involves refining the mesh incrementally and observing the changes in the results. The goal is to reach a point where further refinement does not significantly alter the outcome, indicating that the mesh is sufficiently fine to capture the essential behavior of the model.

Convergence testing is typically done by observing key quantities of interest, such as stress or displacement, and checking their variation with successive mesh refinements. A stable trend in these quantities suggests that the mesh is adequately refined for accurate results.



© 2023 – FEA ACADEMY

# Real-world Examples of Mesh Validation

## Case Studies in Industry

Real-world examples provide valuable insights into the application and importance of mesh validation in various industries. For instance, in the aerospace industry, where safety and precision are paramount, rigorous mesh validation is a standard part of the design process. Detailed case studies often reveal how iterative mesh refinement and quality assessment were used to ensure the integrity and reliability of critical components like wing structures or engine parts.

## Learning from Practical Applications

Another example can be found in the automotive industry, particularly in crash simulations. These simulations involve complex interactions and material behaviors, making mesh validation a crucial step. Case studies show how engineers used error estimation and convergence testing to validate the mesh, ensuring that the simulations accurately predicted the structural response in crash scenarios.

These real-world examples underscore the importance of thorough mesh validation in ensuring the accuracy and reliability of FEA results. They highlight the need for a systematic approach to mesh quality assessment, error estimation, and convergence testing, tailored to the specific requirements and complexities of each project.



© 2023 – FEA ACADEMY

# Section 8. Guidelines for Effective Mesh Generation

This section provides practical guidelines and a step-by-step approach to effective mesh generation in FEA, incorporating best practices for different types of analysis and tips from industry experts. These guidelines are designed to help engineers and analysts create efficient, accurate meshes, leading to reliable and insightful analysis results.

## Step-by-Step Approach for New Projects

### Initial Assessment and Planning

Starting a new FEA project involves an initial assessment of the project requirements. This includes understanding the geometry of the part or assembly, the material properties, the types of loads and constraints, and the expected behavior under these conditions. Planning also involves determining the goals of the analysis: whether it's for validating design concepts, optimizing design, or failure analysis.

### Geometry Preparation and Simplification

The next step is preparing and simplifying the geometry for meshing. This involves removing unnecessary details that won't significantly affect the analysis results, like small fillets and holes, to make the meshing process more straightforward and efficient. It is called “Defeaturing”. However, care must be taken to ensure that these simplifications don’t compromise the accuracy of the model.

### Mesh Generation and Refinement

With the prepared geometry, the next phase is generating the initial mesh. This involves choosing the appropriate element type (1D, 2D, or 3D) and size, based on the complexity of the geometry and the analysis requirements. After the initial mesh generation, it’s essential to refine the mesh in areas where higher accuracy is needed, such as regions with high stress gradients, complex geometries, or around load applications and boundary conditions.



© 2023 – FEA ACADEMY

# Best Practices in Meshing for Different Types of Analysis

## Structural Analysis

In structural analysis, the focus should be on areas of high stress and potential failure. The mesh should be fine enough to capture stress concentrations but not so fine as to make the analysis impractically long. For dynamic analysis, consider the frequency content of the loading and refine the mesh to accurately capture the dynamic behavior of the structure.

## Thermal Analysis

For thermal analysis, the mesh must be capable of accurately modeling heat flow and temperature distribution. This often requires a different meshing strategy than structural analysis, focusing on areas of high thermal gradient. Special attention should be given to the interfaces between different materials, as these can be critical in thermal analysis.

## Fluid Dynamics

In fluid dynamics, the quality of the mesh is paramount, especially near boundaries where the flow behavior is complex. The mesh should be fine enough to capture boundary layers and potential flow separation areas. Adaptive mesh refinement based on flow characteristics can be particularly effective in fluid dynamics simulations.



© 2023 – FEA ACADEMY

# Tips from Industry Experts

## Leveraging Experience and Case Studies

Industry experts often emphasize the importance of learning from past projects and case studies. This can involve understanding how mesh refinement strategies were successfully implemented in similar projects or how specific challenges were addressed.

## Continuous Learning and Adaptation

The field of FEA is continually evolving, with new techniques and technologies emerging regularly. Staying updated with the latest developments in meshing algorithms, software capabilities, and best practices is crucial for effective mesh generation. Experts also recommend regular training and engagement with the FEA community to share insights and learn from others' experiences.



© 2023 – FEA ACADEMY

# Conclusion

## Recap of Key Takeaways

### Essential Role of Meshing in FEA

Throughout this guide, we've emphasized the critical role that mesh generation plays in Finite Element Analysis. A well-constructed mesh is the foundation upon which accurate and reliable FEA simulations are built. From the basics of mesh types and quality assessment to advanced techniques for complex analyses, effective meshing strategies are indispensable for successful FEA outcomes.

### Adaptation to Different Analysis Needs

We've explored how meshing strategies must adapt to various analysis. This adaptability is crucial, as the effectiveness of a mesh is highly context-dependent. The case studies and applications discussed highlight how tailored meshing approaches are employed across different industries, underscoring the mesh's role in addressing specific engineering challenges.

### Common Pitfalls and Best Practices

The guide also delved into common pitfalls in mesh generation, such as over-refinement and inappropriate element choice, and provided practical guidelines to avoid these issues. Best practices, drawn from industry expertise, emphasize the importance of a methodical approach to mesh generation, balancing accuracy with computational efficiency, and the need for continuous learning and adaptation.



© 2023 – FEA ACADEMY

# The Future of Mesh Generation in FEA

## Emerging Technologies and Trends

Looking ahead, the future of mesh generation in FEA is intertwined with advancements in technology. Emerging trends like AI and machine learning are set to revolutionize meshing processes, potentially automating complex decisions and optimizing mesh quality beyond current manual capabilities. These technologies promise to enhance efficiency, reduce human error, and open new possibilities in simulation complexity.

## Integration with Design and Manufacturing

Another future direction is the closer integration of FEA meshing with design and manufacturing processes. As additive manufacturing and digital twins become more prevalent, the need for meshing techniques that can seamlessly interface with these technologies grows. This integration will enable more streamlined workflows from design to analysis to production, enhancing the overall efficiency and effectiveness of engineering projects.

## Adapting to Complex and Multiphysics Analyses

As engineering problems become more complex, involving multiphysics scenarios and advanced materials, the demand for sophisticated meshing capabilities will increase. Future developments in mesh generation must focus on accommodating these complexities, enabling more accurate and comprehensive simulations. The ability to effectively mesh for multiphysics problems will be a critical factor in advancing the capabilities of FEA.



© 2023 – FEA ACADEMY

# Meshless Solutions in Finite Element Analysis

## Introduction to Meshless Methods

Meshless methods, as the name suggests, are techniques in Finite Element Analysis that do not rely on a traditional mesh. These methods, emerging as a significant advancement in computational mechanics, use points scattered throughout the domain to represent the geometry and properties of the model. The appeal of meshless methods lies in their ability to handle large deformations, complex geometries, and highly dynamic problems more efficiently than traditional mesh-based FEA.

## Advantages Over Traditional Meshing

One of the primary advantages of meshless methods is their flexibility in dealing with complex geometries and boundaries. Unlike traditional meshing, which can become exceedingly complex and time-consuming for intricate models, meshless methods can adapt more easily to such variations.

## Challenges and Limitations

Despite their advantages, meshless methods also come with challenges. One of the main limitations is that setting up a meshless simulation can be complex, demanding a deeper understanding of the underlying algorithms and their implementation. There is also the challenge of integrating meshless methods with existing FEA software and workflows, which are predominantly mesh-based.

## Current Applications and Future Potential

Meshless methods are currently used in specialized applications such as fracture mechanics, biomechanics, and fluid dynamics. Their potential for handling problems that are difficult or impossible to address with traditional meshing makes them a valuable tool in the FEA arsenal. As computational power increases and algorithms become more refined, it's expected that the use and capabilities of meshless methods will expand, opening up new possibilities in simulation and analysis.

## Integrating Meshless Solutions into Conventional FEA Practices



© 2023 – FEA ACADEMY

The integration of meshless solutions into conventional FEA practices is an area of ongoing research and development. This integration promises to combine the strengths of both approaches, offering more robust and versatile tools for analysis. Future FEA software might seamlessly transition between mesh-based and meshless methods, choosing the most effective approach based on the problem at hand. Such hybrid approaches could significantly enhance the accuracy, efficiency, and scope of simulations in various fields of engineering and applied sciences.

## Conclusion

In conclusion, this guide has provided a thorough exploration of mesh generation in Finite Element Analysis, covering essential concepts, practical strategies, and industry-specific applications. The future of mesh generation is poised for significant advancements, driven by technological innovations and the evolving needs of engineering analysis. As FEA continues to be an indispensable tool in engineering, the importance of skilled, informed, and innovative meshing techniques will only grow, underlining the need for continuous learning and adaptation in this dynamic field.



© 2023 – FEA ACADEMY

FEA ACADEMY

Courses

The Book



Dominique Madier

FEA Analyst

www.fea-academy.com