SOFTWARE & TECH

Introduction To Finite Element Analysis Using Solidworks Simulation 2019

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Bridging the Gap Between Design and Reality: An Introduction to Finite Element Analysis Using Solidworks Simulation 2019

In the modern world of product design and engineering, the margin for error has shrunk dramatically. Gone are the days when a prototype had to be physically built, tested to destruction, and then re-designed in an expensive cycle of trial and error. Today, engineers hold a powerful tool in their digital arsenal: Finite Element Analysis (FEA). Among the most accessible and deeply integrated FEA packages available is **Solidworks Simulation 2019**. Whether you are a mechanical engineering student, a design engineer looking to validate your work, or a hobbyist building complex assemblies, understanding the basics of FEA within this software is a critical professional skill. This article serves as a comprehensive introduction to Finite Element Analysis using Solidworks Simulation 2019, guiding you through the core concepts, workflow, and best practices for turning your 3D models into data-driven, validated designs.

What is Finite Element Analysis (FEA)?

Before diving into the software itself, it is essential to grasp the fundamental science behind it. FEA is a computational method used to predict how a product reacts to real-world forces, vibration, heat, and fluid flow. The core principle is simple: a large, complex geometry is broken down into millions of smaller, simpler pieces called **elements**. These elements form a mesh that approximates the shape of your part.

By solving complex mathematical equations for each individual element—equations that account for material properties, boundary conditions, and applied loads—the software can predict the stress, strain, displacement, and temperature distribution across the entire assembly. The result is a virtual test that provides deep insight into structural integrity, points of failure, and overall performance before a single physical prototype is ever created.

Why Solidworks Simulation 2019? The Value Proposition

While many high-end FEA solvers exist (such as ANSYS or Abaqus), Solidworks Simulation 2019 offers a unique value proposition: **seamless integration**. Because it is embedded directly within the Solidworks CAD environment, the workflow is incredibly fluid. You do not need to export geometry, repair translation errors, or manage different user interfaces. Your 3D model is the starting point for your simulation, allowing for rapid design iterations. You can change a fillet radius in the CAD model and immediately re-run the simulation to see the impact on stress concentration. This tight integration makes Simulation 2019 an ideal tool for design engineers who are not necessarily simulation specialists but need reliable, actionable data to drive their design decisions.

Key Capabilities of Solidworks Simulation 2019

This version introduces significant improvements in solver speed and ease of use. Key features include:

- Linear Static Analysis: The most common analysis type, calculating stress, displacement, and factor of safety under constant loads.
- Frequency Analysis: Determining natural frequencies and mode shapes to avoid resonance.
- Buckling Analysis: Predicting the critical load at which a slender structure will suddenly collapse.
- Thermal Analysis: Simulating heat transfer through conduction, convection, and radiation.
- Non-Linear Analysis: Handling large plastic deformations and material non-linearities.
- Automated Mesh Controls: The 2019 edition features enhanced curvature-based meshing that automatically refines the mesh around sharp corners and critical features.

The Core Workflow: A Step-by-Step Guide

Using Solidworks Simulation 2019 follows a logical, five-step process. Mastering this workflow is the key to generating accurate and reliable results.

Step 1: Pre-Processing – Setting Up Your Study

The first step is to define the simulation environment. In the Solidworks menu, you activate the Simulation tab and create a **New Study**. You will select the analysis type (e.g., Static) and name your study. This step also involves defining the material properties. Solidworks Simulation 2019 includes an extensive material library pre-populated with mechanical properties like Young's Modulus, Poisson's Ratio, and Yield Strength. Selecting the correct material is crucial because inaccurate material data leads to inaccurate simulation results.

Step 2: Applying Fixtures and Loads

Here, you replicate the real-world environment of your part. **Fixtures**represent how the part is held in place. Common fixture types include:

- Fixed Geometry: Immobilizes all degrees of freedom on a face or edge.
- Roller/Slider: Prevents movement normal to a face but allows sliding.
- Fixed Hinge: Used for cylindrical faces to simulate pin connections.

External Loadsrepresent the forces acting on the part. You can apply forces, pressures, torques, or gravitational loads. The key here is realism. If you over-constrain a part or apply a load that does not match the operational conditions, your simulation might run successfully, but the results will be meaningless. Solidworks Simulation 2019 provides visual indicators (arrows and symbols) to help you verify your setup before solving.

Step 3: Creating the Mesh

The mesh is the heart of FEA. In Solidworks Simulation 2019, you can choose between different element types, but the most common are solid tetrahedral elements. The software offers several mesh controls:

- Standard Mesh: A uniform mesh size across the entire model.
- Curvature-Based Mesh: Automatically refines the mesh in areas with high curvature, such as small fillets or holes.
- Blended Curvature-Based Mesh: An advanced option in 2019 that provides a smoother transition between small and large elements, improving accuracy in complex geometry.

A critical concept here is **mesh convergence**. A very coarse mesh might not capture stress gradients accurately, leading to incorrect results. Conversely, an excessively fine mesh increases computational time dramatically. The best practice is to start with a default mesh, solve, then refine the mesh in high-stress areas and solve again, comparing results until they stabilize.

Step 4: Solving the Problem

Once the pre-processing is complete, you click the "Run This Study" button. Solidworks Simulation 2019 then executes the solver. The Direct Sparse solver handles complex static studies efficiently, while the Large Problem Direct Sparse solver handles very large assemblies with millions of degrees of freedom. The solver writes the equations for each element, assembles the global stiffness matrix, applies the boundary conditions, and solves for displacements. During this process, you can monitor the solution progress and estimated time to completion.

Step 5: Post-Processing – Interpreting the Results

After the solver finishes, the real work begins. The software generates multiple result plots by default. The most important plots include:

- Stress Plot (von Mises): Identifies areas of high stress. You compare this against the yield strength of your material to determine the Factor of Safety.
- Displacement Plot: Shows how much the part deforms under load. Use this to ensure deflections are within acceptable limits.
- Strain Plot: Visualizes the deformation relative to the original geometry.

Solidworks Simulation 2019 allows you to animate the deformation, create cross-section views to see internal stress distribution, and probe specific nodes for exact values. The **Factor of Safety (FOS)**plot is arguably the most valuable for designers. It color-codes the model based on the safety margin; red indicates potential failure, while blue indicates over-design (too much material). The goal of engineering optimization is to achieve a uniform FOS across the part, typically between 1.5 and 2.0, depending on industry standards and safety requirements.

Advanced Workflow Techniques in Simulation 2019

Once you are comfortable with the basic static analysis, Solidworks Simulation 2019 offers advanced capabilities that can further improve your design process.

Contact and Assembly Analysis

Most real-world products are assemblies, not single parts. Simulation 2019 excels at handling contacts between different components. You can define bonded contact (parts are glued together), no penetration contact (parts can touch but not cross), or allow sliding with friction. This is essential for analyzing bolted connections, press fits, or hinged mechanisms.

Design Study and Optimization

The **Design Study**tool allows you to run multiple simulations automatically by varying parameters like dimensions, material, or loads. You can define a goal—such as minimizing mass while keeping the maximum stress below a threshold—and the software will iterate through different design configurations to find the optimal solution. This turns FEA from a validation tool into a powerful design optimization engine.

Thermal Stress Coupling

In many applications, parts are subjected to both thermal and mechanical loads. Simulation 2019 allows you to perform a thermal analysis first to determine the temperature field, and then use that temperature distribution as a load in a subsequent static analysis. This coupled approach is vital for simulating electronic enclosures, engine components, or any part exposed to a hot environment.

Best Practices for Accurate and Efficient FEA

To get the most out of Solidworks Simulation 2019, you must adopt a disciplined approach. FEA is a powerful tool, but it is also easy to misuse. Here are key best practices to follow:

- Simplify Geometry: Remove small features like threads, chamfers, and tiny holes that are not critical to the

structural behavior. These details create massive mesh requirements without significantly affecting the results.

- Validate with Hand Calculations: For simple geometries (like a cantilever beam), calculate the expected deflection using beam theory and compare it to your FEA result. This verifies your setup before you trust results on complex parts.
- Perform Mesh Convergence: Never trust the results from a single mesh. Double the mesh density in critical areas and check if the stress value changes by more than 5%. If it does, refine further.
- Check Boundary Conditions: The most common source of error is over-constraining the model. In reality, parts are not perfectly rigidly fixed. Use appropriate fixtures that allow for realistic deformation.
- Use Symmetry: If your geometry and loading are symmetric, only model one-half (or one-quarter) of the part. This drastically reduces mesh size and solution time.
- Interpret Results Realistically: A sharp corner in your model will produce an infinite stress in theory. The software will show a high stress concentration there, but this is a modeling artifact. Always look at stress in the real geometry (with fillets) and ignore singularities.

Common Pitfalls and How to Avoid Them

Even experienced users can fall into traps. Here are common issues specific to Solidworks Simulation 2019 and how to avoid them:

- Mismatched Units: Ensure your material properties, loads, and geometry are all in consistent units (e.g., SI or Imperial). Solidworks Simulation 2019 provides unit systems, but you must verify them.
- Incorrect Material Assignment: A material with a low yield strength will show failure, while a stronger material on the same part will not. Double-check your material selection.
- Mesh Failure: If the mesh fails to generate, it is often due to very complex geometry or sliver faces. Use the "Defeature" tool to simplify the model.
- Ignoring Large Displacement: If your part deforms significantly (more than 10% of its characteristic dimension), the small deflection assumption is invalid. Enable "Large Displacement" in the solver settings for accurate results.

The Future of FEA and Solidworks Simulation

Solidworks Simulation 2019 represents a mature, highly capable version of the software, but it is essential to recognize that FEA is continuously evolving. Cloud-based simulation, real-time simulation (interactive FEA), and integration with generative design are changing the landscape. However, the fundamental principles taught in this introduction—understanding mesh, boundary conditions, and results interpretation—remain timeless. Mastering these concepts in Solidworks Simulation 2019 provides a solid foundation that will serve you well as the technology advances.

Conclusion

Finite Element Analysis is no longer a niche skill reserved for specialist analysts. With tools like **Solidworks Simulation 2019**, it has become an accessible and indispensable part of the modern design engineer's workflow. By understanding the basic principles of FEA, following the structured workflow of pre-processing, solving, and post-processing, and adhering to best practices like mesh convergence and realistic

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