GENERAL

introduction to finite element analysis using solidworks simulation 2019

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Introduction

When engineers and designers aim to predict how a component will behave under real-world loads, they turn to a powerful tool known as Finite Element Analysis (FEA). Among the many FEA packages available, SolidWorks Simulation 2019 stands out for its integration with the popular CAD platform, intuitive workflow, and robust simulation capabilities. This article provides an **introduction to finite element analysis using SolidWorks Simulation 2019**, guiding you from the basics of FEA to the practical steps of creating, solving, and interpreting a simulation within the 2019 release.

What Is Finite Element Analysis?

Finite Element Analysis is a numerical method that subdivides a complex structure into smaller, manageable elements. By applying mathematical equations to each element, engineers can calculate stresses, strains, deformations, temperatures, and more. The collective behavior of all elements gives insight into how the entire part will perform under specified conditions.

FEA is widely used in aerospace, automotive, consumer products, and many other industries because it reduces the need for costly prototypes and allows rapid iteration of designs.

Why SolidWorks Simulation 2019?

SolidWorks Simulation 2019 offers several advantages:

- Seamless CAD Integration – Your SolidWorks model is the starting point; no file conversion is required.
- User-Friendly Interface – The Simulation tab provides a guided workflow that even beginners can follow.
- Advanced Meshing – Automatic and manual meshing options ensure accurate results.
- Wide Range of Physics – From linear static to thermal, transient, and nonlinear analyses, 2019 covers most typical engineering scenarios.
- Optimized Performance – Improved solver speed and parallel processing capabilities.

Getting Started: Setting Up Your First Simulation

1. Preparing the Model

Before launching a simulation, clean up your SolidWorks part:

1. Remove any unused sketches or features.
2. Ensure all dimensions are defined and the model is fully constrained.
3. Apply appropriate material properties to each part or component.
4. Check for gaps or overlaps that could cause meshing issues.

SolidWorks 2019 encourages the use of “Component” and “Assembly” features to organize complex designs, which simplifies the application of boundary conditions later.

2. Launching SolidWorks Simulation

Navigate to the **Simulation** tab on the Command Manager. Click **New Study** and choose the type of analysis you need—most beginners start with a **Static Linear** study for structural problems.

3. Defining Materials

In the Study Manager, right-click **Materials** and select **Assign Material to Part**. SolidWorks Simulation 2019 includes a comprehensive material library (steel, aluminum, composites, plastics). For custom materials, use the **New Material** option and input density, Young's modulus, Poisson's ratio, and thermal properties.

4. Applying Boundary Conditions

Boundary conditions mimic real-world constraints:

- Fixed/Fixed Support – Rigidly restrain all degrees of freedom.
- Pin – Restrict translation but allow rotation.
- Force/Moment – Apply loads.
- Pressure – Apply distributed loads over surfaces.
- Thermal Conditions – Specify temperature or heat flux.

Use the **Apply** button to select faces, edges, or vertices and define the magnitude and direction of loads.

5. Meshing the Model

Meshing is critical: a coarse mesh may miss stress concentrations, while an excessively fine mesh increases computation time. In SolidWorks Simulation 2019, you can:

1. Use the default Automatic Mesh for quick results.
2. Refine mesh in regions of interest by selecting faces and applying Local Mesh Control.
3. Choose element types: tetrahedral for complex geometry, hexahedral for high-accuracy in straight sections.
4. Set element size based on the geometry's feature size and expected stress gradients.

After meshing, review the mesh quality metrics—skewness, aspect ratio—to ensure reliable results.

6. Running the Solver

Click **Run**in the Study Manager. SolidWorks Simulation 2019's solver automatically selects the appropriate algorithm based on the study type. During execution, monitor the progress bar and solver logs for any warnings.

7. Post-Processing Results

Once the solution converges, explore the results:

- Stress (Von Mises, Max Principal) – Identify critical points.
- Displacement – Check for excessive deformations.
- Strain – Useful for fatigue analysis.
- Thermal plots – Temperature distribution and heat flux.

Use contour plots, arrows, and animation to visualize deformations and loads. Export graphs or create screenshots for reports.

Advanced Features in SolidWorks Simulation 2019

Nonlinear Analysis

For problems involving large deformations, material nonlinearity, or contact interactions, the **Nonlinear Static** study type is available. Enable **Large Displacement** and **Material Nonlinearity**in the study settings. Contact pairs can

be defined between mating surfaces to simulate friction or sliding.

Thermal-Mechanical Coupling

SolidWorks Simulation 2019 supports coupled analyses where temperature changes affect mechanical behavior (e.g., thermal expansion). Set up a **Thermal-Structural** study, assign thermal loads first, then link temperature to the structural study.

Optimization Tools

Use the **Design Table** and **Parametric Study** features to perform design optimization. Vary dimensions or material properties across multiple runs, and use the **Optimization** add in to minimize weight while maintaining strength.

Multibody Dynamics

For dynamic simulations involving moving parts, SolidWorks Simulation 2019 includes a **Multibody Dynamics** study. Define joints, apply inertial loads, and observe motion over time.

Best Practices for Reliable Results

1. Validate Mesh Quality – Inspect skewness, aspect ratio, and element size distribution.
2. Use Symmetry – If your model is symmetric, simulate only half to reduce computational load.
3. Apply Loads Consistently – Ensure forces and constraints are applied in the correct coordinate system.
4. Check Solver Convergence – Look for residuals and adjust solver settings if convergence stalls.
5. Document Assumptions – Record material properties, boundary conditions, and any simplifications.
6. Iterate Design – Use simulation feedback to refine geometry before building prototypes.

Real-World Example: Analyzing a Simple Bracket

Consider a rectangular aluminum bracket used in a consumer electronics enclosure. The bracket must support a 10 /kg load without exceeding 0.5 /mm deflection.

Steps:

1. Model the bracket in SolidWorks with appropriate fillets.
2. Assign aluminum (Al 6061) material.
3. Apply fixed support at the bracket base.
4. Apply a downward force of 98 /N at the top center.
5. Mesh with a local refinement at the load point.
6. Run a linear static study.
7. Check the Von Mises stress (should be below yield strength of ~276 /MPa).
8. Verify displacement (should be

Adjust the bracket thickness or add a support rib if results exceed limits. Repeat until design criteria are met.

Learning Resources for SolidWorks Simulation 2019

- Official SolidWorks Documentation – Comprehensive guides and tutorials.
- Online Courses – Platforms like Udemy, LinkedIn Learning, and Coursera offer courses specifically for SolidWorks Simulation.
- Community Forums – The SolidWorks Forum and Eng-Tips host discussions and solutions.
- YouTube Channels – Channels such as “SolidWorks Tutorials” provide step-by-step videos.
- Books – “SolidWorks Simulation for Engineers” by William H. Brown offers in-depth coverage.

Common Pitfalls and How to Avoid Them

Overlooking Material Properties

Using generic or incorrect material data can skew results. Always double-check density, modulus, and Poisson's ratio.

Inadequate Meshing

Skipping local mesh refinement near load points or stress concentrations leads to inaccurate stress predictions.

Misapplied Boundary Conditions

Applying a fixed support where a pin is intended will artificially stiffen the model. Verify the degrees of freedom each constraint allows.

Ignoring Solver Settings

Default solver tolerances may be too loose for high-accuracy projects. Adjust residual and displacement tolerances if necessary.

Conclusion

SolidWorks Simulation 2019 empowers engineers to predict component behavior accurately, streamline design iterations, and reduce physical prototyping costs. By following a structured workflow—preparing the model, assigning materials, applying boundary conditions, meshing wisely, running the solver, and interpreting results—you can harness the full potential of FEA. Whether you're tackling a simple bracket or a complex automotive part, the tools and best practices discussed here provide a solid foundation for successful simulations.

With a solid understanding of **finite element analysis using SolidWorks Simulation 2019**, you're equipped to turn virtual insights into real-world performance.

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