SIMULATION 101 FOR NON-CAE EXPERTS

Mechanical engineering for product development breaks down into three main areas: design, simulation, and analysis. Non-CAE experts, including managers and engineers, can find simulation difficult to comprehend as it involves complex topics, specialized equations, and advanced matrix mathematics.

Simulation is the use of computer models to predict the response of parts, assemblies, and drawings to validate their performance in their operating environments—and then optimize those objects for the best possible solution.

Analysis and simulation tools are critical for product development organizations because they help reduce costs, weight, and time to market. Without simulation and analysis, teams must build and test expensive prototypes, potentially many times.

This eBook will give non-analysts a deeper understanding of the tasks and outputs of the simulation process, including the nature of simulation in product development. Read on to learn more about:

- Basic simulations: Structural, modal, and thermal
- Advanced analyses: Dynamic, mechanisms, and buckling
- Democratizing Finite Element Analysis (FEA) for designers
- Multiphysics and Computational Fluid Dynamics (CFD)

Are you ready to take your first steps into a larger world? Let's go!

Part 1: Types of Simulation for Managers

It's common for managers in product development to not have a simulation and analysis background. Maybe they came from another technical field like electrical engineering or computer science. Or maybe they were program managers with a non-technical background. And that's okay. No one is expected to be an expert or even familiar with the fields of the people they work with. Below is a guide to help managers who are unfamiliar with product-development simulation understand the basics of the craft. Let's take a look at three core analysis types: structural, modal, and thermal

Structural Analysis

A professor at MIT said, "Mechanical engineering is the study of failure. And some of you will become extremely acquainted with that." It's true. Product designers develop concepts for products and then analyze and test them to see if they will survive – or fail – their operating environments.

To simulate those conditions, engineers study something called stress. Structural analysis allows us to calculate stress. When things are tough, people might say, "I'm under a lot of stress." We might also say, "I'm under a lot of pressure." In mechanics, stress and pressure are closely related. Both pertain to force per unit area. They are measured in the same units, like pounds per square inch or Newtons per square millimeter.

When a product is used in its operating environment, it will often experience forces, known as loads. Those loads will either be in tension (pulling), compression (pushing), or both at the same time. Objects in tension will get longer; objects in compression will get shorter. This change in length is known as displacement.

Those loads and displacements result in stress. When the stress is below the yield strength, the object will resume its original shape when the load is removed. When the stress reaches or exceeds the yield strength, the object will be permanently deformed when the load is removed. When the stress exceeds the ultimate tensile strength or the ultimate compressive strength, the object will break.

If the stress on the product is below failure, we can calculate the margin of safety or factor of safety. If the product is overdesigned, we might make it lighter and thinner so that it just meets its requirements. This is optimization.

Modal Analysis

Modal analysis is the study of the frequencies and shapes at which an object will vibrate. If someone plucks a guitar string or a backyard laundry line, it vibrates. You might even see standing waves in the string. That's an example of the object's natural frequency and mode shapes.

Physical objects have not only a natural frequency, but subsequent higher frequencies at which they vibrate. The natural frequency and subsequent frequencies are the object's modes. Product designers need to know a product's modes to avoid exciting it. If energy is pumped into an object at or near one of its modes, it will vibrate more and more (resonate), until it can literally tear itself apart. For example, if a pump is rotating at a certain frequency, you would not want to mount any components to it near that frequency.

The higher an object's natural frequency, the stiffer the object is. Modal frequencies can help product designers choose between different design alternatives. Modal analyses can also be used as inputs into more advanced analyses, like dynamic analyses, which will be covered in the next section.

Thermal Analysis

It's probably obvious that "thermal" refers to temperature. Thermal analysis studies the results of loads or boundary conditions that affect the transfer of heat to or from an object. As heat is applied, an object expands. As something cools, it contracts. As discussed earlier, when length changes ("displacement"), that induces stress in the object.

There are three different ways in which temperature can be transferred to or from an object:

- Conduction: This is direct contact. If you have ever touched a hot stovetop, a laptop case near its battery, or a car's hood after a long drive, that's conduction.
- Convection: This is heat transfer via a fluid, like air or water. If you have ever cooled yourself off by sitting in front of a fan, you have benefitted from convection.
- Radiation: This is heating an object via electromagnetic waves. You use radiation to heat leftovers or pop popcorn in a microwave oven.

Engineers perform thermal analysis to calculate the final temperature (also known as "steady state"), the temperature over time ("transient thermal"), or how fast heat is entering or exiting an object ("heat flux").

Part 2: Advanced Simulation Analyses

Let's take a look at three more classes of analyses. These are a bit more advanced than those discussed in the previous section, but still commonly used. They are:

- Dynamic analyses for loads that change as a function of time or frequency
- Mechanism analyses for machines with moving components
- Buckling analyses for thin sections under compression

Dynamic Analysis

Dynamic analyses study the effects of loads that can change, as a function of time or frequency, in the operating environment. This contrasts with static analyses, where the loads are constant and we want to know stress or displacement at equilibrium (when everything settles). For example, a static analysis is how much a parked car's roof deflects when a bike rack and two bicycles are mounted onto it.

Common types of dynamic analyses include:

- Dynamic Time. In these studies, the load changes over time and the goal is to generate results for how the stress, displacement, or other measures change over time. For example, if a submersible vehicle travels down to 1000 meters over 5 minutes, the hydrostatic pressure changes over time. To see the effects on the structure over that time, perform a dynamic time analysis.
- Dynamic Frequency. In many machines with components that rotate or oscillate, loads are applied and removed over a certain time, called the period. The frequency is one divided by the period. For example, when a car's tachometer reads 3000 rpm (revolutions per minute), the engine's crankshaft is revolving 50 times per second. That is a frequency of 50 Hz (Hertz); the period is 0.02 seconds. To study the stress on the crankshaft from the connecting rod and piston, perform a dynamic frequency analysis.
- Random vibration. Driving over a bumpy country road or flying in an airplane can be bouncy. How shaky or turbulent it will be, or the exact times and frequencies of the jostling, is unknown. However, in certain operating environments, it's possible to measure how much bouncing will probably occur.Analysts can generate a graph that represents the probability of the jostling (accelerations) experienced as a function of frequency. This is called a Power Spectral Density (PSD) or more accurately, an Acceleration Spectral Density (ASD) plot. These probabilities can be used as inputs to the random vibration analysis to get a sense of the likelihood that a product can survive that environment.

CREO SIMULATION ENHANCEMENTS

Mechanism Analyses

Mechanism analyses study the behavior of machines, which are assemblies of components in which some parts can move. In machines, moving parts can translate, rotate, or some complex combination of translation and rotation. Advanced connections between components, like cams, gears, belts, and pulleys, can also be simulated.

Mechanism analyses can be divided into two main categories: kinematics and dynamics. Kinematics is the study of motion. In a kinematic analysis, engineers are typically interested in the position, velocity, and/or acceleration of the moving components.

Dynamics is the study of forces. In a mechanism dynamics analysis, real world entities like springs, dampers, gravity, and external forces can be incorporated. In addition to position, velocity, and acceleration, forces and reactions in the mechanism's component can also be measured. These results can also be used as inputs—load cases—to structural analyses. Mechanism analyses can check for collision between parts. Sometimes it's essential to make sure parts don't collide, like the spinning blades in a lawn mower housing. In other mechanisms, the intended outcome is one component pushing another in the next part of a sequence. For example, the braking system of a car moves pads or shoes to slow down wheels using friction.

Motion envelopes that represent the space through which components move during the analysis can also be generated.

Buckling Analysis

When a thin section is under compression, it can yield or fail below the material's yield strength or ultimate compressive strength because of a special failure mode called buckling. A buckling analysis allows us to study this special case.

For example, take a drinking straw and stand it on its end on a tabletop. Then press down on the straw. The straw is an example of a thin section; the thickness is small compared to its length. By pressing down on it, it is being compressed. At some point, it will usually bend sharply and quickly somewhere in the middle. This is buckling.

The main results of the buckling analysis are:

- **The buckling load factor. Multiplying the static load by this factor yields the critical load when the object can become unstable.**
- **The buckling mode shape, which indicates how and where the object is likely to move at the critical load.**

Buckling analyses are often performed for structural housings and sheet metal components.

Now that we have covered basic and advanced analyses, let's examine the math and science behind how Finite Element Analysis performs a simulation.

Part 3: Democratizing FEA for Designers

So far, we've reviewed some of the major analysis types. In this section, let's go over a basic understanding of the way Finite Element Analysis (FEA) works.

Classical Analysis vs. FEA

What is called "classical mechanics" uses "closed form equations" to calculate displacement, reaction forces, stresses, and more. "Closed form" means that a person plugs in various quantities related to the shape of the object, material properties, loads, and so on, in order to get the answer. These closed form equations are available from handbooks like Roark's Formulas for Stress and Strain.

Unfortunately, these closed form equations only work for specific situations, like beams, bars, flat plates, pipes, and pressure vessels. If someone wants to analyze modern products with complicated geometry, like an engine block, the housing for personal electronics, medical equipment, and so on, closed form equations don't exist.

There is one equation necessary to understand FEA, and that is the spring force equation:

$$F=k\times x$$

It states that the amount of Force (F) in a spring is equal to the spring constant (k) times the displacement (x) of the spring (how much it is compressed or extended).

Finite Element Analysis takes a "divide and conquer" approach. It breaks up models with complex shapes into much smaller pieces. These can be tetrahedra (pyramids), wedges (prisms), and bricks (blocks). These small pieces are the finite element part of FEA. Equations for the spring constants, or stiffness, of these elements do exist.

The process of breaking up a complex shape into all these smaller elements is known as "meshing." Analysis software puts the elements together into what's called a "stiffness matrix."

Classical Analysis vs. FEA

When setting up a simulation, the forces applied to the model are defined. Since the stiffness matrix and the forces are known, we should be able to solve for the displacements in the spring equation above.

The problem with these matrices is that they are too complex to solve; there are more variables than the number of equations. However, when setting up a simulation, constraints are defined. These specify where the model is held down and kept from moving. At these locations, the displacement is zero. These constraints reduce the equations down to a solvable number. In this way, FEA solves for the displacements of the elements.

Calculating Stresses

Now that we know the displacements, it's possible to calculate the change in each element's length compared to the original length. This proportion is the strain.

This quantity is useful because a lot of materials, especially metals, have a linear relationship between the strain and the stress. By multiplying this strain by a material property called Young's modulus, you get the stress.

Materials have a value called the yield strength. If loads to objects below their yield strength are applied, the object will deform; remove the load, and the object will return to its original shape. If loads above the yield strength are applied, the object will be permanently deformed. Most structural analyses are valid only up to the yield strength.

Putting It All Together

- Apply material properties to parts
- Define constraints for where the models are kept from moving
- Define loads that mimic the real-world conditions for the operating environment
- Mesh the model
- Set up the analysis and run the simulation
- Analyze the results

The math behind FEA is very complicated. Simulation is both an art and a skill. It takes knowledge, training, and experience for an analyst to generate good results, but this is a baseline overview of the process. Finding out what your products experience in their environments is known as design validation. Once you perform that, you can optimize your models to find the best possible solution.

If performing simulation analyses and validating designs on your own seems daunting, there are software tools that can help. There are two main simulation tools in the PTC Creo portfolio—Creo Ansys Simulation and Creo Simulation Live—for use throughout the development process to reduce physical prototyping costs, design cycle times, and analysis department backlog.

Creo Ansys Simulation (CAS) seamlessly integrates the power of Ansys, the leader in engineering simulation, directly into Creo. Built specifically for designers and engineers, this easy-to-use, fully-featured, high-fidelity simulation tool leverages Ansys' capabilities for thermal, structural, and modal analyses.

With Creo Simulation Live (CSL), you get instant feedback on your design decisions in the CAD environment as you work. This ground-breaking technology performs structural, thermal, and modal analysis on 3D CAD designs in seconds. Created especially for design engineers, the analysis updates dynamically in real-time as you edit, create new features, or change properties. You can iterate fast, incorporating as many design changes as you want, confident that you've made the best-informed decisions possible.

Learn more about CAS, CSL, and the differences between both in this short video.

>>> THE CREO ADVANTAGE:

Creo is the 3D CAD solution that helps you accelerate product innovation to build better products faster. Easy-to-learn Creo uses a model-based approach to seamlessly take you from the earliest phases of product design to manufacturing and beyond. Combining powerful, proven functionality with new technologies including generative design, real-time simulation, advanced manufacturing, IIoT and augmented reality, Creo helps you iterate faster, reduce costs and improve product quality. Creo is also available as a SaaS product, providing innovative cloud-based tools for real-time collaboration and streamlined license management and deployment. The world of product development moves quickly, and only Creo delivers the transformative tools you need to build competitive advantage and gain market share.

To learn more, contact us for a follow up demo.

Language Support English, German, French, Japanese, Russian, and Simplified Chinese