UNLOCK NEXT-LEVEL PRODUCTIVITY

Ansys Discovery is the first simulation-driven design tool to combine instant physics simulation, proven Ansys high-fidelity simulation and interactive geometry modeling in a single user experience.

Accelerate time to market and reduce product development costs

Build better products with less physical prototyping and lower production costs

Create market-leading products that set you apart from the competition

Discovery Overview

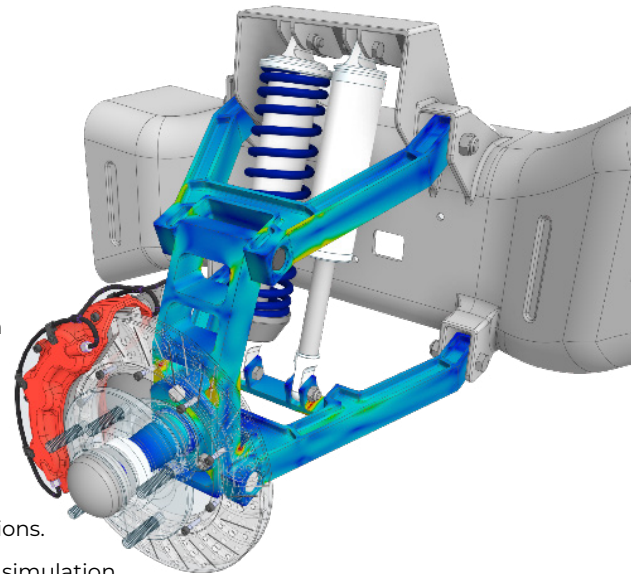**Ansys Discovery reveals critical insights early in the design process**

By combining interactive geometry modeling and multiple simulation capabilities in a first-of-its-kind product, Discovery allows designers and engineers to answer critical design questions early in the product development process. This upfront approach to simulation saves time and effort on prototyping as you explore multiple design concepts in real time with no need to wait for simulation results.

- **First-of-its-kind GPU-based live physics simulation**
- **Radically easy-to-use interface**
- **Embedded multiphysics simulation**
- **High-fidelity simulation powered by Ansys solvers**

Uplevel Your Product Design

An intuitive and easy-to-use interface allows design engineers to test multiple design ideas quickly with live physics simulation, then refine design ideas with simulation results backed by Ansys high-fidelity solvers. Power cross-team collaboration in Discovery with direct transfer to Ansys Mechanical and Fluent simulation software to simulate more complex physical behaviors.

- **CAD Connected** – Work with geometry from virtually anywhere.
- **Automated Prep** – Quickly prepare models for simulation and create design variations.
- **Design Exploration** – Rapidly exploring multiple design concepts with live physics simulation.
- **Cross-Collaboration** – Get help from simulation experts with model transfer to other Ansys applications.

"Discovery equips our team with a much better understanding of the physics behind our products very early in the design process, enabling them to meet customer requirements more precisely, avoid overengineering and eliminate uncertainties. This has resulted in improved product performance, increased design efficiency and shortened product development cycles."

Stefan Macho
Head of R&D Simulation, HAWE Hydraulik

/ Capabilities

Geometry Modeling - Import any CAD or neutral geometry format and quickly cleanup, simplify and parameterize geometry for simulation. Easily create or edit geometry within the entire simulation workflow for rapid design iterations, and rapidly convert faceted data to surfaces or solids for downstream use. A Discovery license can also be used to run Ansys SpaceClaim for even more modeling capabilities.

Fluid Simulation - Rapidly evaluate flow rate, pressure drop, fluid temperature and fluid loads for both steady-state and transient fluid and fluid-thermal simulations with instantaneous interactive results or high-fidelity simulation via the Fluent solver.[1]

Topology Optimization -Easily optimize the shape of designs and improve strength to weight ratio including a variety of manufacturing constraints with instantaneous interactive topology optimization for generative design. Rapidly convert the optimized shape into smooth CAD geometry to validate designs.
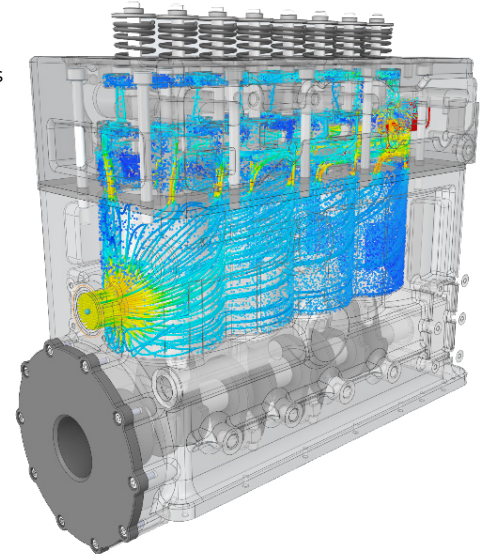
Stress Analysis - Rapidly evaluate deformation and stress on components and assemblies including a variety of contact conditions for realistic assembly modeling with instantaneous interactive results or high-fidelity simulation via the Mechanical solver.[1]

Modal Analysis
Easily evaluate natural frequency and vibration characteristics of parts and assemblies with instantaneous interactive results or high-fidelity simulation via the Mechanical solver.[1]

Thermal Simulation - Rapidly evaluate temperature, heat transfer and thermal-stress of components and assemblies for both steady-state and transient thermal simulation with instantaneous interactive results or high-fidelity simulation via the Mechanical solver.[1]

Transfer to Fluent or Mechanical - Rapidly transfer simulation data from Discovery to either Fluent or Mechanical for advanced physics simulation to enable collaboration between design engineers and simulation analysts.

[1] Additional product license required to access the Mechanical or Fluent solver.

ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
ansysinfo@ansys.com

If you've ever seen a rocket launch, flown on an airplane, driven a car, used a computer, touched a mobile device, crossed a bridge or put on wearable technology, chances are you've used a product where Ansys software played a critical role in its creation. Ansys is the global leader in engineering simulation. We help the world's most innovative companies deliver radically better products to their customers. By offering the best and broadest portfolio of engineering simulation software, we help them solve the most complex design challenges and engineer products limited only by imagination.

Visit www.ansys.com for more information.