

# Mechanical Overview

Realize your Product Promise through superior simulation technology. Customers trust our accurate solutions to obtain insight into real-world product performance, driving innovation and business success.



**Help meet product code regulations**



**Improve product performance and reliability**



**Root cause failure analysis**



**Reduce time and cost of product development**

## / Mechanical Overview

**Insight into Real-World Product Performance** - Ansys Mechanical is a best-in-class finite element solver with structural, thermal, acoustic and piezoelectric simulation capabilities. Static analysis accurately predicts the deformed shape, steady-state temperature and stress distribution. Transient analysis helps you to understand dynamic effects of loading. Linear dynamics analysis reveals vibration characteristics.

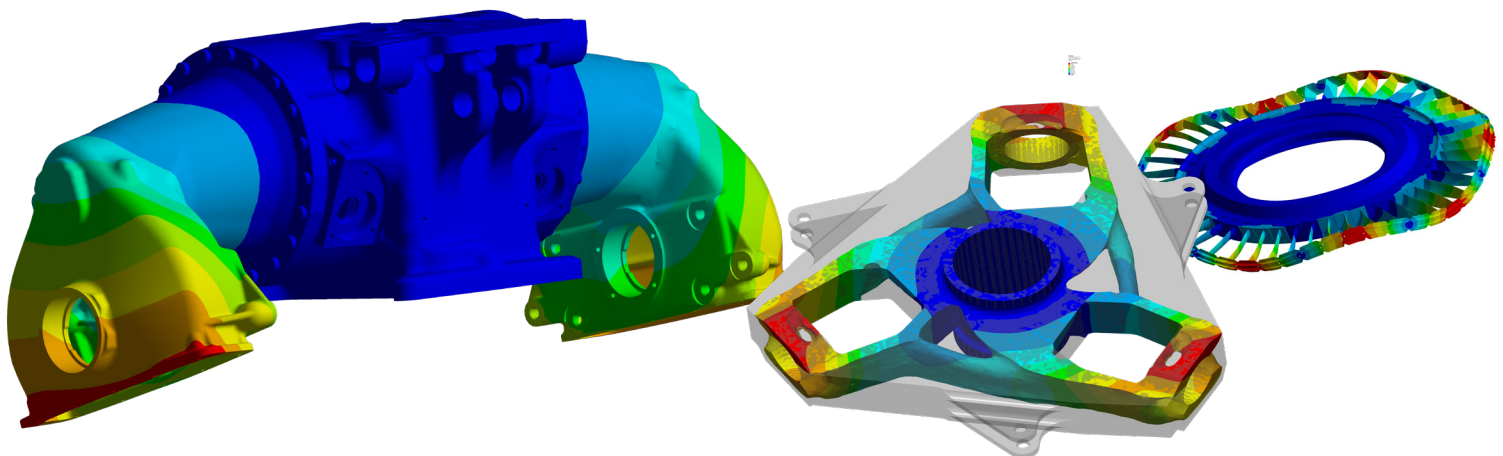
**Combine Solver Accuracy and Speed** - Superior solver accuracy and speed provide trustworthy information and enable enormous productivity gains through scalable HPC performance.

**A Parametric, Multiphysics Platform** - Ansys Workbench enables robust connection to commercial CAD tools, providing clickbutton design point updates. Seamlessly integrated multiphysics capabilities are available with our fluids and electrical solvers.

## / Mechanical is Your Solution

Ansys Mechanical is a dynamic environment that has a complete range of analysis tools from preparing geometry for analysis to connecting additional physics for even greater fidelity. The intuitive and customizable user interface enables engineers of all levels to get answers fast and with confidence.

- **CAD connected - work with geometry from anywhere**
- **Parametric - vary geometry and loads to find the optimal solution**
- **Quick to learn - intuitive, modern user interface**



## CAPABILITIES

<b>ELEMENTS</b>	1D, 2D and 3D continuum and structural elements, rigid body representation
<b>MATERIAL MODELS</b>	Linear and nonlinear elastic, viscoelastic, rate-dependent and rate-independent plasticity, geomechanical, gaskets, composites
<b>INTERACTION MODELS</b>	Linear and nonlinear contacts, joints
<b>FRACTURE MODELS</b>	SMART, cohesive zone, XFEM
<b>ADVANCED SOLVER TECHNOLOGIES</b>	Nonlinear adaptivity, cyclic symmetry, submodeling, substructuring (CMS), inverse analysis, multidimension modelling
<b>ANALYSIS TYPES</b>	Static, transient dynamics, modal, response spectrum, harmonic, random vibration, acoustics, explicit dynamics, hydrodynamics, multibody dynamics
<b>USER PROGRAMMABLE FEATURES</b>	Custom material models, interactions, loads, elements
<b>MULTIPHYSICS</b>	Coupled element technology and one to two way connection to other solvers (eg. fluid-structure interaction, electromagnetics)
<b>HIGH PERFORMANCE COMPUTING</b>	Paralleled solvers for faster solution times

## TOOLS

<b>GEOMETRY PREPARATION TOOLS</b>	Ansys Spaceclaim, Ansys DesignModeler
<b>DESIGN OF EXPERIMENTS</b>	Ansys DesignXplorer, Ansys OptiSLang
<b>COMPOSITES PRE AND POST PROCESSING</b>	Composites PrepPost
<b>MATERIALS DATABASE</b>	Ansys Granta Material Data for Simulation
<b>MATERIALS HOMOGENIZATION</b>	Ansys Material Designer
<b>TOPOLOGY OPTIMIZATION</b>	Level set and SIMP methods, shape optimization
<b>RIGID BODY DYNAMICS</b>	Ansys Rigid Body Dynamics, Ansys Motion
<b>HYDRODYNAMICS ANALYSIS</b>	Ansys AQWA
<b>EXPLICIT DYNAMICS ANALYSIS</b>	Ansys Explicit, Ansys LS-DYNA, Ansys Autodyn
<b>FATIGUE ANALYSIS</b>	Ansys nCode DesignLife
<b>SOLVE MANAGEMENT</b>	Remote Solve Manager, Distributed Compute Service

Ansys, Inc.  
www.ansys.com  
ansysinfo@ansys.com  
866.267.9724

© 2020 ANSYS, Inc. All Rights Reserved.