ANSYS®

12.0

12.0 RELEASE

Features

Structural Analysis
- Static, modal, harmonic and transient analysis
- Spectrum analysis
- Buckling analysis
- Random vibration
- Geometric, material, and contact nonlinearities
- Displacements transferred to thermal, electric, magnetic or fluid analysis

Thermal Analysis
- Steady-state and transient analysis
- Conduction, convection and radiation
- Phase change
- Mass transport
- Fluid elements
- Temperature dependent material properties
- Temperatures transferred to structural, electric, magnetic or fluid analysis

Electrostatic Analysis
- Charge-based electric elements
- Trefftz method for open domain
- Electrostatic forces transferred to structural analysis

Steady-State Current Conduction
- Current-based electric elements
- Infinite elements for open domain
- Currents transferred to magnetostatic analysis
- Resistive losses transferred to thermal analysis

Low-Frequency Electric Field Analysis
- Charge and current based elements
- Infinite elements for open domain
- Time-harmonic and time-transient quasistatic
- Resistive and dielectric losses transferred to thermal analysis
- Currents transferred to magnetic analysis

Comprehensive Multiphysics

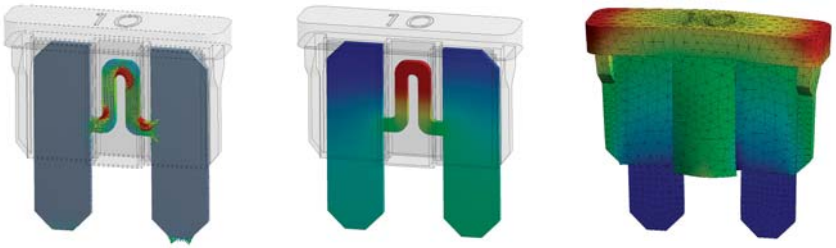
In an expanding range of applications, engineers and designers must be able to accurately predict how complex products will behave in real-world environments where multiple types of coupled physics interact. ANSYS multiphysics solutions allow our customers to evaluate their designs operating under real-world multiphysics conditions. Software from ANSYS enables engineers and scientists to simulate the interaction between structural mechanics, heat transfer, fluid flow and electromagnetics all within a single, unified engineering simulation environment.

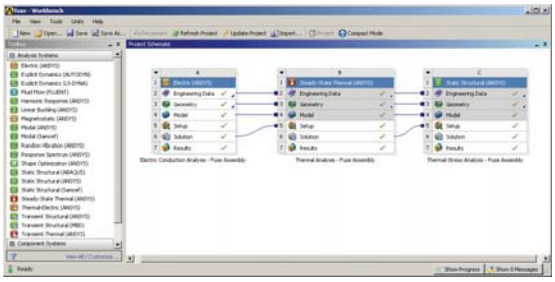
Built on Proven Solver Technology

ANSYS multiphysics solutions are built on proven solver technology validated by many years of application in the world's leading universities and corporations. Technical depth and breadth in all physics is essential to understanding the complex interactions between different physics disciplines. Our industry-leading solver technology for all physics disciplines, in conjunction with the engineered scalability of the ANSYS product portfolio, allows users to solve challenging, real-world multiphysics problems.

Unified Simulation Environment

The ANSYS® Workbench™ platform is a powerful multi-domain simulation environment that harnesses the core physics from ANSYS, enables their interoperability and provides common tools for interfacing with CAD, repairing geometry, creating meshes and post-processing results. The ANSYS Workbench environment enables multiphysics simulation within an open and adaptive software architecture.





The ANSYS Workbench platform is a powerful multiphysics simulation environment. The project schematic shows the multiphysics workflow for a coupled electric conduction, heat transfer and subsequent thermal-stress analysis of an automotive blade fuse.

ANSYS®

Features

Magnetostatic Analysis
- Magnetic vector potential and scalar potential elements
- 3-D edge flux element formulation
- Resistive losses transferred to thermal analysis
- Magnetic forces transferred to structural analysis

Low-Frequency Magnetic Analysis
- Magnetic vector potential elements
- 3-D edge flux formulation
- Quasistatic magnetic
- Time-harmonic analysis for linear materials
- Time-transient analysis for linear and nonlinear materials
- Both permeable and saturable materials
- Permanent magnets
- Resistive and eddy current losses transferred to thermal analysis
- Magnetic forces transferred to structural analysis

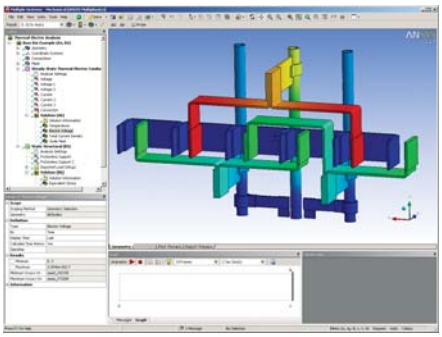
High-Frequency Electromagnetic Analysis
- First- and second-order tangential vector elements
- 3-D brick, pyramid, prism and tetrahedral element shapes
- Cavity modal analysis
- Harmonic analysis: wave propagation, radiation and scattering
- Isotropic and anisotropic materials
- SPICE equivalent circuit output
- Resistive and dielectric losses transferred to thermal analysis

Circuit Analysis and Coupling
- Static, harmonic and transient
- Coupled electromagnetic field analysis and discrete electric circuits
- Resistors, capacitors, inductors, diodes, transformers, voltage and current sources
- Electromechanical transducer
- Interactive circuit builder
- Coupling to both stranded and massive conductors

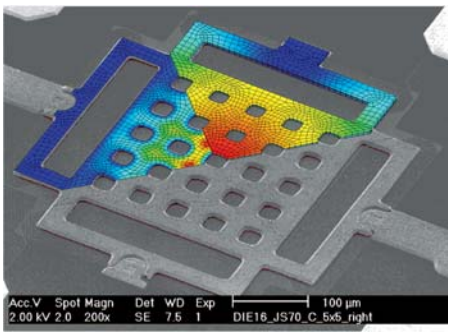
Ion Optics
- Charged-particle tracing in electric or magnetic static fields, or both
- Plot trajectories in 2-D or 3-D

Flexible Simulation Methods

ANSYS multiphysics solutions deliver two proven solution techniques to solve multiphysics problems — the direct coupled-field elements and the ANSYS Multi-field solver. These approaches provide flexible simulation methods to solve a broad range of both direct and sequentially coupled multiphysics problems such as induction heating, electrostatic actuation, Joule heating and fluid structure interaction (FSI).



Buss-bar of a short-circuit test transformer with current up to 150 kA – thermoelectric model with temperature-dependent material properties solved in the ANSYS Workbench environment.
Model courtesy of WEG Electrical Equipment.



Coupled electrostatic-fluid-structural model showing pressure (left quadrant) and displacement (right quadrant) overlaid on scanning electron microscope image of an RF MEMS switch.
Image courtesy of EPCOS NL and Phillips Applied Technology.

Direct Coupled-field Elements

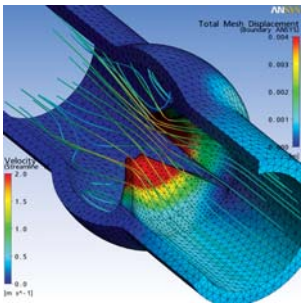
The direct coupled-field elements allow users to solve a coupled-physics problem by employing a single finite element model with the appropriate coupled-physics options set within the element itself. A direct coupled-field solution simplifies the modeling of multiphysics problems by allowing users to create, solve and post-process a single analysis model for a wide variety of multiphysics problems.

Key Features of Direct Coupled-field Elements

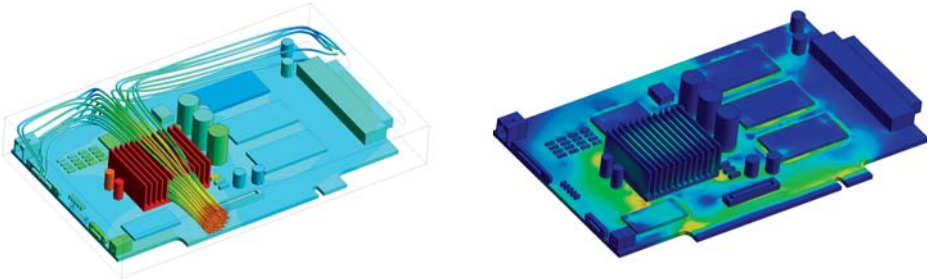
- Single model simplifies multiphysics simulation
- Support for a broad array of coupled physics
- Robust for highly nonlinear multiphysics solutions
- Support for parallel processing
- Includes nonlinear geometric effects

ANSYS Multi-field Solver

The ANSYS Multi-field solver enables our customers to solve multiphysics problems by using automated implicit sequential coupling, which couples multiple single-physics models in one unified simulation. The ANSYS Multi-field solver employs robust, iterative coupling in which each physics discipline is solved sequentially and convergence is obtained between the individual disciplines at each time point during the solution. The multi-field coupling is based on customized interprocess communication technology and no third-party coupling software is required.



Fluid structure interaction of a three-lobe valve, simulation solved using the ANSYS Multi-field solver. The model includes Non-Newtonian blood flow and anisotropic hyperelasticity to model the biological tissue.



A conjugate heat transfer solution and subsequent thermal-stress analysis of a computer graphics card. Fluid streamlines and solid temperatures (left) and thermal stresses (right) are shown for the coupled simulation.

Key Features of the ANSYS Multi-field Solver

- Automated implicit sequential coupling
- No third-party coupling scheme required
- Automated mesh morphing for nonstructural elements
- Support for a dissimilar mesh interface between physics models
- Allows collaboration between physics experts
- Advanced FSI

Benefits of Multiphysics Solutions from ANSYS

ANSYS, Inc. continues to lead the CAE industry in the development of multiphysics solutions that provide the high-fidelity simulations required to meet the challenges of today's demanding product development requirements. ANSYS multiphysics solutions offer a portfolio of software that provides analysts with powerful simulation tools for solving industry's toughest multiphysics challenges. Features include:

- Superior solvers for all physics simulation
  - Structural mechanics, heat transfer, fluid flow and electromagnetics
- Flexible multiphysics simulation built on proven solver technology
- A unified simulation environment for multiphysics analysis
- Fully parametric analysis allows design of experiments, robust design and design optimization for multiphysics solutions
- Parallel scalability for multiphysics analysis
- World class support and services from ANSYS

Features

Fluid Flow Analysis

- Tetrahedral, hexahedral, prism and/or pyramid elements
- Steady-state and transient flow
- Laminar and turbulent flows
- Incompressible, compressible – subsonic, transonic, supersonic
- Rotating or stationary frame of reference
- Conjugate heat transfer
- Radiation
- Newtonian and non-Newtonian fluids
- User-defined equations and species transport
- Free surface modeling
- Fluid structure interaction
- Fluid pressures and temperatures transferred to structural analysis
- Heat flux and temperatures transferred to thermal analysis

Acoustics

- Modal, harmonic and transient analysis
- Fluid medium
- Fully coupled fluid-structural

Direct Coupled-field Elements

- Piezoelectricity
- Piezoresistivity
- Piezocaloric effect
  - Thermoelastic damping
- Coriolis effect
- Electroelasticity
- Thermoelectricity
  - Joule heating, Peltier, Seebeck and Thomson effects
- Thermal-structural
- Thermal-electric-structural

Multi-field Solver

- Electrostatic-structural
- Electrostatic–structural-fluid
- Thermal-structural
- Thermal-electric
- Thermal-electric-structural
- Thermal-electric-fluid
- Thermal-fluid
- Electromagnetic-thermal
- Electromagnetic-structural
- Fluid structure interaction

Optimization

- Design optimization
- Topological optimization
- Probabilistic design
- Variational technology
- Parametric simulation

ANSYS®

12.0 RELEASE

Features

ANSYS Parametric Design Language
- Macros
- Parametric modeling
- If-then-else constructs
- Do-loop features
- Array parameters
- Array parameter operations
- Trigonometric functions

Solvers
- Sparse direct
- Jacobi conjugate gradient (JCG)
- Incomplete Cholesky conjugate gradient (ICCG)
- Pre-conditioned conjugate gradient (PCG)
- Quasi-minimal residual (QMR)
- Algebraic multigrid (AMG)
- Eigensolvers
  - Block Lanczos
  - PCG Lanczos
  - Supermode modal solver
  - Householder (reduced)
  - Unsymmetric
  - QR-damped

ANSYS Supported Platforms*
- HP-UX® 64
- HP-UX Itanium® 64
- IBM® AIX® 64
- Sun® Solaris™ 64
- Sun Solaris x64
- Linux® 32
- Linux Itanium 64 (RedHat® 4,5)
- Linux x64 (EM64T/Opteron 64)

ANSYS Workbench Supported Platforms*
- Windows® AMD64/EM64T
- Windows XP
- Windows Vista

Additional Modules
- ANSYS® DesignModeler™
- ANSYS® DesignXplorer™
- ANSYS® Fatigue™
- ANSYS Mechanical HPC
- ANSYS Rigid Dynamics
- ANSYS® Explicit STR™

* Refer to www.ansys.com for a current list of supported hardware platforms and operating systems

Multiphysics Product Spectrum

Multiphysics solutions from ANSYS are offered through a range of products and provide the depth and breadth of capabilities to meet current requirements as well as a seamless upgrade path for future simulation needs.

ANSYS® Multiphysics™ software offers a comprehensive product solution for both multiphysics and single-physics analysis. The product includes structural, thermal, fluid and both high- and low-frequency electromagnetic analysis. The product also includes solutions for both direct and sequentially coupled physics problems including direct coupled-field elements and the ANSYS Multi-field solver.

ANSYS® Mechanical™/Emag software offers a comprehensive solution for structural, thermal, and low-frequency electromagnetic analysis. The product also includes solutions for both direct and sequentially coupled physics problems including direct coupled-field elements and the ANSYS Multi-field solver for supported physics.

ANSYS Mechanical/CFD-Flo software offers a comprehensive solution for structural, thermal and fluid analysis. The product also includes solutions for both direct and sequentially coupled physics problems including direct coupled-field elements and the ANSYS Multi-field solver for supported physics.

ANSYS Mechanical software offers a comprehensive product solution for structural and thermal analysis. The product also includes solutions for both direct and sequentially coupled physics problems including direct coupled-field elements and the ANSYS Multi-field solver for supported physics.

ANSYS® CFD™ offers a comprehensive product solution for computational fluid dynamics, which includes both the ANSYS® CFX® and ANSYS® FLUENT® solvers. The product also includes both one-way and two-way FSI when combined with an ANSYS Mechanical license.

The ANSYS Advantage

With the unequalled depth and unparalleled breadth of ANSYS engineering simulation solutions, companies are transforming their leading-edge design concepts into innovative products and processes that work. Today, 97 of the top 100 industrial companies on the "*FORTUNE* Global 500" invest in engineering simulation as a key strategy to win in a globally competitive environment. They choose ANSYS as their simulation partner, deploying the world's most comprehensive multiphysics solutions with engineered scalability that delivers the flexibility to solve their most complex engineering challenges.

www.ansys.com

GSA Contract Holder

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

Toll Free U.S.A./Canada:
1.866.267.9724
Toll Free Mexico:
001.866.267.9724
Europe:
44.870.010.4456
eu.sales@ansys.com

MKT0000302 4-09