

ANSYS Explicit Dynamics Solutions

Analyzing a product's response and identifying potential problems before it is manufactured can mean the difference between the product's failure or success. Getting it right the first time is especially important for products that experience severe conditions such as impacts, penetrations, explosions and other similar loadings during use.

ANSYS explicit dynamics engineering simulation solutions are ideal for simulating physical events with severe loading that occur in a short period of time and may result in material damage or failure. These types of events are often difficult or expensive to study experimentally. Simulation provides insight and a detailed understanding of the fundamental physics taking place and gives engineers a chance to make necessary changes before their products are put into service, when mistakes in design can be costly.

Based on first principles, ANSYS explicit dynamics products solve the conservation equations of mass, momentum and energy using explicit time integration. ANSYS® Explicit STR™, ANSYS® AUTODYN® and ANSYS® LS-DYNA® solutions collectively offer the full range of capabilities needed to address complex explicit problems.

ANSYS Explicit STR

ANSYS Explicit STR software is the only structural program that is fully integrated into a unified environment capable of incorporating a range of multiphysics solutions, including electrical, thermal, mechanical and computational fluid dynamics (CFD).

The solution method used by ANSYS Explicit STR software is based on the robust and time-tested Lagrange solver of the ANSYS AUTODYN analysis program.

Unified Environment

ANSYS Explicit STR technology enables users to complete their analysis significantly faster than with any other tool in the industry. Competition and economic conditions demand innovative, reliable products that can be designed, manufactured, tested and delivered to customers quickly. At the same time, manufacturing and material costs continually need to be reduced. ANSYS Explicit STR software operates within the ANSYS® Workbench™ platform, which provides a seamless interface to CAD geometry for analysis, automatic meshing and parametric design optimization; users can leverage their expertise without needing to deal with arcane details of simulation methods. Used within the ANSYS Workbench platform, ANSYS Explicit STR software gives users the distinct advantage of being able to complete their analyses quickly, allowing manufacturers to get better quality products to market faster.

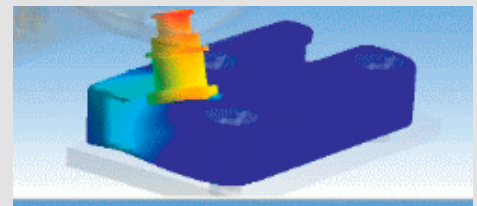
ANSYS Explicit STR and the ANSYS Workbench Environment

ANSYS Explicit STR software uses the same graphical user interface (GUI) as the widely popular ANSYS® Mechanical™ solver and all other analysis systems in the ANSYS Workbench environment. This means that, for customers who have used any of the ANSYS Workbench analysis systems, the learning curve for using an explicit solution will be significantly reduced, further enabling the use of this technology in a timely manner.

Industry Solutions



Sport equipment design



Window closure mechanism

Courtesy of MACO.



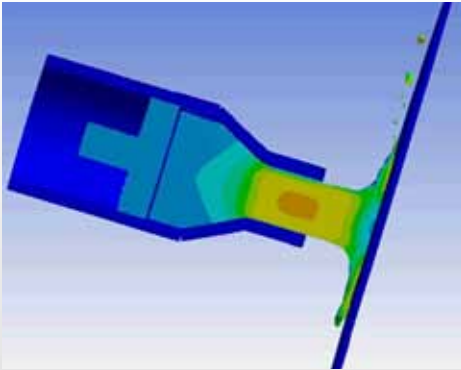
Stability of a bottle

Industry Solutions

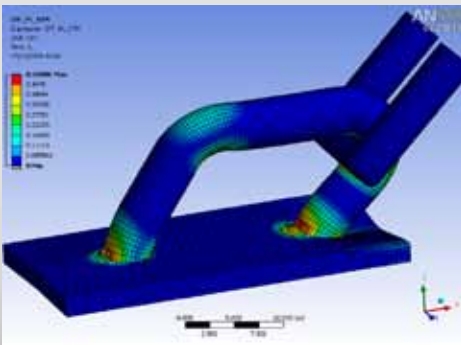
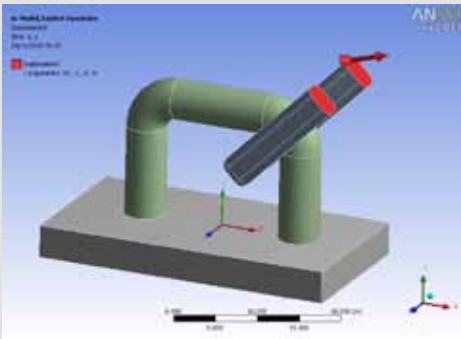


Crimping of wires

Courtesy of TYCO Electronics.



Pressurized water jet impacting a deformable wall



Lifting lug under tension, impending failure

Applications

The ANSYS Explicit STR tool is well suited for solving:

- Drop tests, such as those found in electronics and consumer goods industries
- Low- to high-speed impacts, such as those found in industries from sporting goods to aerospace
- Highly nonlinear plastic buckling events that occur in many manufacturing processes or in the use of products
- Material degradation and failure, such as those seen in defense and homeland security
- Applications that require breakable contact, such as adhesives or spot welds often found in electronics and automotive industries
- Structural problems involving complex contact and interactions

Problem Setup

A key benefit of ANSYS Explicit STR software is the workflow advantage of operating within the ANSYS Workbench environment. While many different simulation processes are possible, below is an example of the typical steps a user might take:

- Select materials to be used from the ANSYS Workbench native Engineering Data library
- Generate geometry in ANSYS® DesignModeler™ software or use the bidirectional, parametric connections to all major CAD systems to import geometry into the ANSYS Workbench framework
- The explicit preference option automatically selected by powerful ANSYS meshing solutions creates a mesh with reasonably uniform element sizes that is well suited for explicit analysis
- Automatically create part-to-part contact with the body interactions tool
- Fine-tune contact specifications if desired by utilizing breakable bonds to connect parts
- Load and/or support an assembly and/or parts as usual
- Solve interactively either in the background or via remote solution manager (RSM)
- With a single mouse click, reproduce all of these steps if the CAD model is changed
- View progress of a solution in real time using the concurrent post-processing capability available in the ANSYS Workbench environment.
- Explore alternative design ideas via parametric changes to the CAD model or material properties and produce new results automatically
- Use the ANSYS Design Exploration capability to automate the parametric model space exploration

ANSYS AUTODYN

ANSYS AUTODYN software is an advanced explicit analysis tool for modeling complex nonlinear dynamics involving solids, fluids, gases and their interactions. It is a powerful, multiphysics explicit program that provides a wide range of capabilities for simulating problems involving severe loadings in a short period of time, such as high-speed impacts and explosions.

ANSYS AUTODYN software offers multiple solution methods including Lagrange, Euler, arbitrary Lagrangian-Eulerian (ALE) and smoothed-particle hydrodynamics (SPH), a mesh-free solver. ANSYS AUTODYN software is fully integrated with an intuitive, easy-to-use GUI that is further enhanced with all of the features available in the ANSYS Workbench platform. These features enable users to easily and quickly prepare models from CAD geometry, create a mesh well suited for explicit analysis, and set up multiple calculations using parameters based on geometry, details, material models and initial conditions.

Applications

ANSYS AUTODYN software is ideal for solving:

- Hypervelocity impacts useful in space vehicle designs
- Oil well perforations for enhanced extraction
- Rock drilling under extreme conditions
- Meteorite, asteroid and comet impact analyses
- Aircraft-impact risk assessments for critical structures
- Blast loadings on structures
- Drop test simulations
- Characterizations of materials subjected to highly dynamic loadings
- Designs of protective armor
- Personnel carrier protection
- Explosive decommissioning of structures

ANSYS AUTODYN software tightly integrates the pre-processing, post-processing and analysis modules for maximum productivity. The software can be run in serial or parallel mode on the Microsoft® Windows® operating system as well as on Linux® systems. Both shared-memory and distributed-cluster systems are supported.

Compelling Features

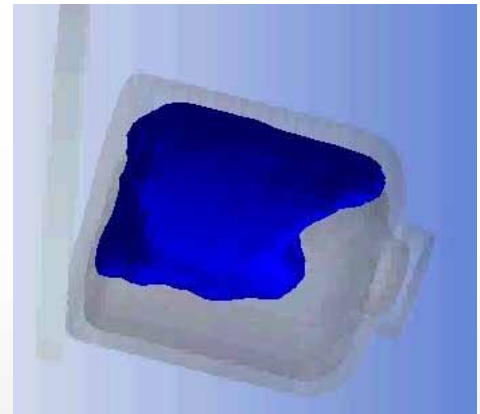
ANSYS AUTODYN software has been continually developed over the last 25 years to naturally and effectively handle the nonlinear behavior of fluids and structures in an integrated fashion.

A unique and powerful capability of the program is its ability to handle fluid structure interaction (FSI) and sophisticated material models in the structure and fluid part of a problem.

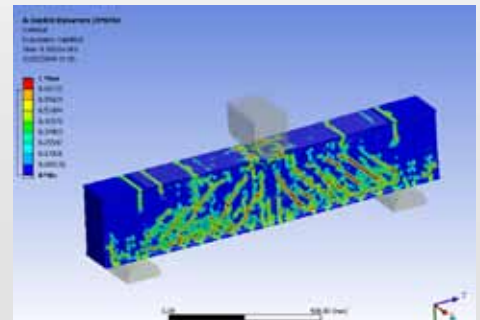
ANSYS AUTODYN capabilities exceed all other explicit programs because of key features that include:

- Integrated and coupled response of fluids, structures and materials
- Multiple solvers, including those for structural analysis, fluid flow, gas flow and SPH
- The ability to combine all solution methods in a single problem enabling the most accurate and efficient solution possible
- Finite element (FE) solver for structural analysis
- Finite volume solvers for fast, transient fluid- and gas-flow analysis
- SPH/particle methods for large deformation and fragmentation
- Material models and data for metals, ceramics, glass, concrete, soils, explosives, water, air and many other solids, liquids and gases
- Full-featured material models with easy-to-modify equation of state (EOS), strength model and failure model
- Statistical failure models to simulate natural fragmentation
- Analysis of materials with strength — such as metals — in addition to fluids and gases in all solvers
- Comprehensive remapping capabilities from structural to fluid flow and vice versa
- Interactive GUI with easy-to-use visualization
- Interactive simulations seamlessly integrated with pre- and post-processing
- Extensive material model library combining thermodynamic and constitutive responses
- Serial and parallel computation on shared-memory and distributed-memory systems
- Extensive correlation with physical experiments confirmed by the ANSYS AUTODYN user community with extensive publications
- Extensive set of tools to enhance accuracy and reduce computational time

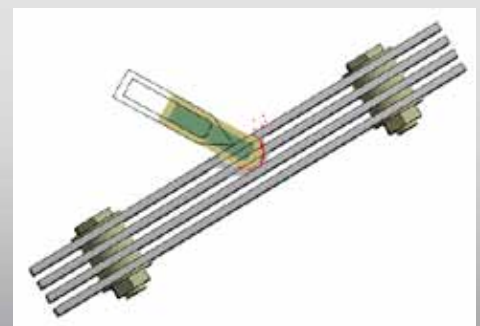
Industry Solutions



Drop test of a bottle containing water

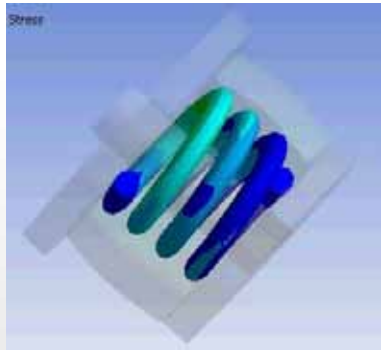


Reinforced concrete beam with loading resulting in failure

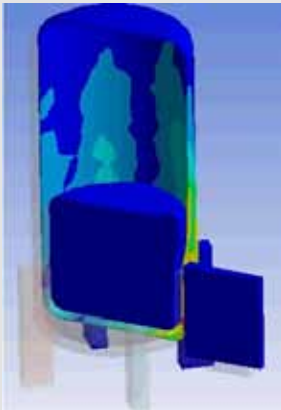


Projectile impacting and penetrating multilayered armor

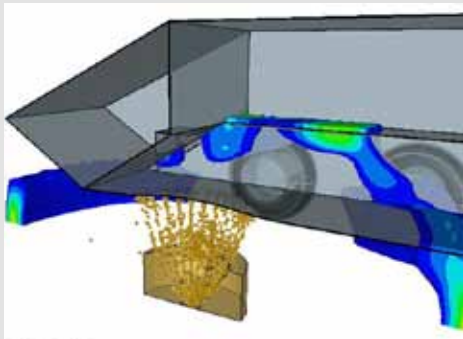
Industry Solutions



Shock absorber spring showing complex contact



Pressure vessel containing a liquid being impacted



Land mine detonating under a generic armored vehicle resulting in loading from blast and soil throw

ANSYS LS-DYNA

ANSYS LS-DYNA technology is the result of a collaborative effort between ANSYS, Inc., and Livermore Software Technology Corp. (LSTC). Introduced in 1996, the capabilities and robustness of ANSYS LS-DYNA software have helped customers in numerous industries resolve highly intricate design issues. The powerful explicit solutions available in the LS-DYNA program have been available to ANSYS Mechanical users from within the ANSYS Parametric Design Language (APDL) environment. The LS-DYNA solver is available within the ANSYS Workbench platform with all of the compelling features, including CAD interfaces and automatic meshing. Input files for LS-DYNA (.k files) can be created using the same GUI that is offered in the ANSYS Mechanical program and all other analysis programs from ANSYS.

Short duration, solid-to-solid dynamic events with medium to severe loadings are well suited for the use of ANSYS LS-DYNA technology. Applications, such as impact, blade containment, stamping and forming, present challenges that are ideal for simulating with ANSYS LS-DYNA software.

ANSYS LS-DYNA software supports both 2-D and 3-D explicit finite elements and features a comprehensive set of single-surface, surface-to-surface and node-to-surface contacts including the automatic creation of contact on surfaces. The software also offers optional methods for fast solution processing. Symmetric multiprocessing (SMP) and massively parallel processing (MMP) methods optimize the performance of the program with the use of multiple CPUs and cores to deliver explicit simulation results in a fraction of the time of a single CPU.

The material library for ANSYS LS-DYNA software offers constitutive material models that range from simple and computationally fast to more sophisticated models for virtually all materials that may need to be modeled with an explicit solver.

ANSYS multiphysics solutions are available for customers using ANSYS® Structural™ and ANSYS Mechanical solutions. The two solutions can be combined in the ANSYS APDL environment with ANSYS LS-DYNA software for problems that benefit from both implicit and explicit solutions involving pre-stressing and spring-back, such as stamping operations.

The ANSYS Advantage

With the unequalled depth and unparalleled breadth of engineering simulation solutions, companies are transforming their leading-edge design concepts into innovative products from ANSYS and processes that work. Today, almost all 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 to solve their complex engineering challenges. The engineered scalability of solutions from ANSYS delivers the flexibility customers need, within an architecture that is adaptable to the processes and design systems of their choice. No wonder the world's most successful companies turn to ANSYS — with a track record of 40 years as the industry leader — for the best in engineering simulation.

ANSYS®

www.ansys.com

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



Contract Holder