

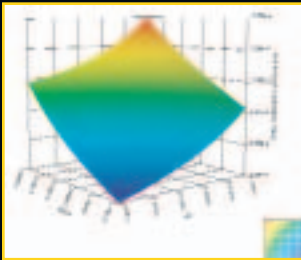
Various software modules augment and enhance the capabilities of standard ANSYS license configurations.

Variational Technology Modules

ANSYS provides two separately licensed Variational Technology modules for ANSYS:

DesignXplorer VT – Add this module to any ANSYS license supporting structural analysis and create a response surface for FEA results in linear elastic static structural or normal modes analyses. This works within the Workbench Environment to select geometric CAD parameters or geometric parameters from ANSYS DesignModeler or material properties and thicknesses. The module works within ANSYS to select material properties, real constant values (e.g. thickness) and collection of elements to be active or inactive.

ANSYS Frequency Sweep Module VT – Included with every ANSYS Emax license, this module performs a frequency sweep for high-frequency electromagnetics analyses much faster than traditional methods. The module also may be added to any ANSYS Multiphysics license.



Capabilities

- ▶ Supports analyses that are structural static normal modes analyses with linear elastic isotropic materials
- ▶ Works with sparse solver
- ▶ Parameters that can be varied include:

Discrete Variation of Elements:
LINK180, SHELL181, PLANE182
PLANE183, SOLID185, SOLID186
SOLID187, BEAM188, BEAM189
COMBIN14

Material Properties Variation of Elements:
LINK180, SHELL181, PLANE182
PLANE183, SOLID185, SOLID186
SOLID187

Element Property Variation of Elements:
COMBIN14, MASS21, SHELL181

Variational Technology

Look beyond traditional aspects of simulation and analyze entire systems within simulated environments with ANSYS DesignXplorer VT, a robust Variational Technology solution that gives users a broader view of design concepts.

When combined with the Workbench Environment, this powerful application allows designers and analysts to make intelligent design directives given multiple competing objectives. These changes can include geometric CAD parameters as well as material properties and thicknesses. Both linear elastic static stress and normal modes analyses are supported.

When combined with the ANSYS Environment, this powerful application allows analysts to gain detailed understanding of their designs. DesignXplorer VT's proprietary Variational Technology offers the capacity to study, quantify and graph the structural analysis response to alternative design parameters for both linear elastic static stress and normal modes analyses.

Using ANSYS Variational Technology to automatically calculate all of the necessary derivatives within a single finite element solution, DesignXplorer VT allows users to perform increasingly quick and accurate what-if scenarios to periodically test and verify design ideas. A traditional design of experiments method would require many solutions to characterize the behavioral changes due to parameter variations.

With Variational Technology, users can approach product design decisions much more efficiently. Depending on the analysis problem, Variational Technology can provide acceleration factors between 10 and several thousand.

Meshing Enhancements

A Hex dominant option has been added to the Element Shape Control, allowing an unstructured hex mesh to be applied to solid bodies. This functionality is enabled by the Advanced Structural Meshing module license that is purchased separately. Hex dominant meshing offers benefits mainly when meshing bodies with large amounts of interior volume, or bodies that transition from sweepable parts. It offers minimal benefits when meshing thin complex parts (like a cellular phone case) or when meshing sweepable bodies.

Accessible from within the Workbench Environment, the Hex Dominate meshing currently only supports higher order elements. The hexahedral elements can be converted to lower order elements within the ANSYS Environment.



Hex dominant mesh of a crankshaft.

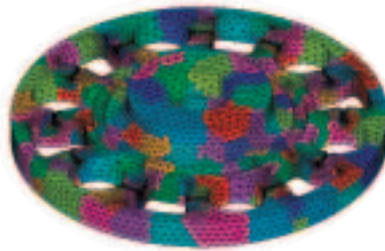


Hex dominant mesh of a machine cover.

Parallel Performance for ANSYS Overview

Multiprocessing computer environments (consisting of multiprocessor servers or networked workstation clusters) may be employed to generate simulation and analysis results much more quickly. The Parallel Performance module facilitates this highly effective operation. The module adds two solvers to any ANSYS license supporting the ANSYS Environment. This add-on module features four solvers that permit engineers to affordably resolve large and complex engineering problems many times faster than before. The solvers are:

- Algebraic Multigrid (AMG) Solver
- Distributed Domain Solver (DDS)
- Distributed Pre-conditioned Conjugate Gradient (DPCG) - New at 8.0!
- Distributed Jacobi Conjugate Gradient (DJCG) - New at 8.0!



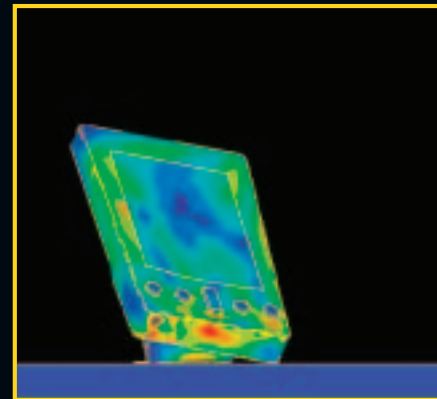
ANSYS 8.0 includes two new distributed solvers in addition to the DDS solver: the DPCG and the DJCG solvers. The DPCG and the DJCG solvers are faster, more robust, solve a wider range of analyses, and require much less memory than the DDS solver. All three solvers require the Parallel Performance for ANSYS add-on module.

Algebraic Multi-Grid (AMG) Solver

- Intended for shared-memory multiprocessing servers.
- Scales up to eight processors yet maintains excellent performance on as few as two.
- Utilizes an overall stiffness matrix procedure for pre-conditioning of linear and nonlinear static and full transient structural analyses.
- Ideal for ill-conditioned models resulting from large aspect ratios or models with shell and/or beam elements attached to solid elements.

Distributed Domain Solver (DDS)

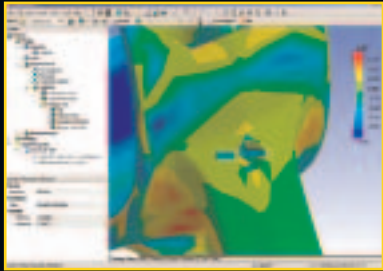
- Scalable solver designed for either a distributed-memory environment or a combined distributed/shared memory environment.
- Divides models into multiple contiguous finite element groupings (between 1,000 and 10,000 degrees of freedom each) extracted from the complete model.
- May be scaled to double-digit processors, even over a network.



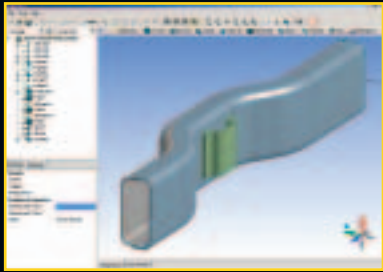
Above:
Three frames from a Personal Desktop Assistant (PDA) drop test performed in ANSYS LS-DYNA with the Drop Test Module added. The initial impact (top) was calculated as being dropped from a height of 1,829 mm (approximately six feet). The secondary impact (middle) exhibits separation of the PDA's batteries from the main housing. The bottom image displays the rebound of the secondary impact exhibited by the device in the simulation.

The ANSYS Workbench Environment represents the next step in the continuing evolution of the ANSYS scalable solution. First introduced in ANSYS DesignSpace 6.0, this GUI offers users more intuitive simulation operational controls.

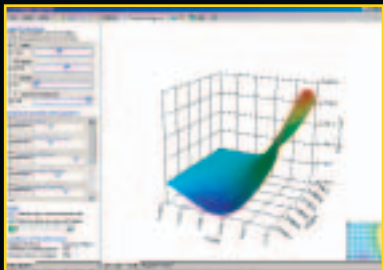
Fatigue – Adds the capability to simulate performance under anticipated cyclic loading conditions over anticipated product life span. Contour plots display fatigue life, damage, factor of safety, stress bi-axiality and fatigue sensitivity.



DesignModeler – Aids in refining CAD geometry for simulation. Provides users with a solution for implementing analysis-led design rather than CAD-led design.



DesignXplorer – This module adds advanced parametric control to permit study of the simulation response to proposed modifications. The module includes the ability to quantify and graph results.



Distributed Pre-conditioned Conjugate Gradient (DJCG) and Distributed Jacobi Conjugate Gradient (DPCG) Solvers

Two new, faster distributed memory solvers, the Distributed Pre-conditioned Conjugate Gradient (DPCG) solver and the Distributed Jacobi Conjugate Gradient (DJCG) solver, are now available in the Parallel Performance for ANSYS add-on module. These solve a wider range of analyses and require less memory than the DDS solver, and are available in the Parallel Performance for ANSYS add-on.

- DPCG: Based on an enhanced theory for the PCG solver, the DPCG solver targets distributed parallel processing. The DPCG solver preserves all of the merits of the PCG solver and can be run on either shared memory or distributed memory machines with superior scalability to the PCG solver.
- DJCG: Based on an enhanced theory for the JCG solver, the DJCG solver targets distributed parallel processing. Scalability of this solver is superior to the JCG solver with little additional memory required.

Drop Test

Product drop tests present users with unique parameters for simulation and analysis. The Drop Test module further enhances an ANSYS LS-DYNA license by adding a complement of loads and conditions tailored specifically for such analyses.

Simulation Applications

- ▶ Consumer electronics
- ▶ Household appliances
- ▶ Children's toys
- ▶ Lawn and garden equipment
- ▶ Power tools
- ▶ Sporting goods
- ▶ Shipping containers and contents

Loads and Conditions

- ▶ Automatic target surface generation
 - Angle of incline
- ▶ Initial velocities
 - Translational
 - Angular
- ▶ Frictional effects
- ▶ Drop heights
- ▶ Gravity

Results Evaluation

- ▶ Animation output
- ▶ Stress
- ▶ Deflection
- ▶ Strain
- ▶ Motion
- ▶ Windows AVI format output
- ▶ Time history graphs
- ▶ Displacement vs. time
- ▶ Velocity
- ▶ Acceleration

User Interface

- ▶ Intuitive
- ▶ Single dialogue box
- ▶ Drop test-specific documentation

Based on Design of Experiments (DOE), DesignXplorer works from within the Workbench Environment to perform DOE analyses of any Workbench simulation, including those with geometric CAD parameters. Although it requires many more analyses to be performed, and is typically slower than DesignXplorer VT, DesignXplorer is not limited to linear elastic structural analyses. DesignXplorer can be used with ANSYS Parametric Design Language (APDL)-based files to perform DOEs on existing or new ANSYS analyses.