MSYS®

Technical Overview



Table of Contents

The ANSYS Finite Element Analysis Program	1
Evolution of the ANSYS Program	1
Overview of the Program	1
User Interface.	
Graphics	
Processors	
Database	
File Format	
Reducing Design and Manufacturing Costs with ANSYS FEA	5
Program Availability	
ANSYS Family of Products	
Example Used I hroughout I his Brochure	
A Note About this Brochure	9
Preprocessing	
Solid Modeling	
Meshing	
Parametric Definition	
Direct Generation of Models	
Solution	
Equation Solver	14
Structural Static Analysis	
Structural Dynamic Analysis	
Transient Dynamic	
Modal	
Harmonic Response	
Response Spectrum	
Random Vibration	
Structural Buckling Analysis	20
Linear Buckling	
Nonlinear Buckling	
Structural Nonlinearities	99
Material Nonlinearities	
Ceometric Nonlinearities	
Flement Nonlinearities	
Static and Dynamic Kinematic Analysis	
Thermal Analysis	
Steady-State	
Transient	
Phase Change	
Thermal-Structural	

Electromagnetic Field Analysis	
Static Electromagnetic Fields	
Time-Varying Electromagnetic Fields	
Electric Field Analysis	
Electric Current Conduction	
Electrostatics	
Electric Circuit Analysis	
Fluid Flow Analysis	
Computational Fluid Dynamics	
Acoustics	
Coupled-Field Analysis	
Piezoelectric Analysis	
Substructuring	
Submodeling	
Material Properties	
The ANSYS Element Library	
P-elements	
Element Table	
Postprocessing	
The ANSYS General Postprocessor	
Time-History Results Postprocessor	
ANSYS Parametric Design Language	
Parameters	
Array Parameters	
Branching and Looping	04 65
Macros	
User Routines	
Design Optimization	
Third-Party Program	
The ANSYS CAD Relations Program	
Quality Assurance	
Customer Services.	
Index	

The ANSYS Finite Element Analysis Program

For over a quarter century, customers have relied on the ANSYS[®] program to help them bring quality products to market quickly. ANSYS, Inc. supports the ongoing development of innovative technology and delivers flexible, enterprise-wide engineering systems that enable companies to solve the full range of analysis problems, maximizing their existing investments in software and hardware.

What started as a one-man operation has grown into an organization of over 200 employees with more than 50,000 users worldwide. Dr. John Swanson founded ANSYS, Inc. in 1970 with a vision to commercialize the concept of computer-simulated engineering, establishing himself as one of the pioneers of finite element analysis (FEA). His work helped spark the beginning of the computer-aided engineering (CAE) industry. Today, many major, multinational corporations have standardized on ANSYS software. ANSYS customers also include the top ten industrial corporations from the *Global Fortune* 100.

ANSYS, Inc. continues its role as a technological innovator. Pioneering breakthroughs include first analysis on the personal computer (PC), first integrated computational fluid dynamics (CFD) capabilities, and first multiphysics capabilities on the PC. ANSYS, Inc. invests in research and development to ensure customers that the ANSYS family of products will grow and change to meet their engineering needs.

ANSYS, Inc. supports a process-centric approach to design and manufacturing, allowing users to avoid expensive and time-consuming "build and break" cycles. In a continuous and collaborative engineering process, on-going analysis is used throughout product development, and everyone works together as a team in their areas of expertise and responsibility. As a strategic partner, ANSYS, Inc. works with companies to manage change and help them stay ahead of the competition. ANSYS analysis and simulation tools give customers ease of use, data compatibility, multiplatform support, and coupled-field multiphysics capabilities.

Evolution of the ANSYS Program

ANSYS has evolved into a multipurpose design analysis software program, recognized around the world for its many capabilities. The first release of the ANSYS program looked much different than it does today; offering only heat transfer and linear structural analysis. It was a batch program, like most in its day, and ran only on a mainframe computer.

The early 1970s brought many changes to the program as the ANSYS staff incorporated new technology and user requests. Nonlinearities, substructures, and a wider assortment of elements were added. The company began looking at the then-new minicomputers and vector graphics terminals. Within a few years, these hardware advancements would pave the way for ANSYS software to move into a new era of CAE.

The interactive mode of operation was a significant addition to the program in the late 1970s. With it, model generation and results evaluation (pre and postprocessing) were greatly simplified. Interactive graphics could be used to verify model geometry, materials, and boundary conditions before an analysis was run. Graphics displays of the analysis results could then be produced immediately for interactive verification.

Today, the program is extremely powerful and easy to use. Each release hosts new and enhanced capabilities that make the program more flexible, more usable, and faster. In this way, ANSYS helps engineers meet the pressures and demands of the modern product development environment.

Overview of the Program

The ANSYS program is a flexible, robust design analysis package. The software operates on major computers and operating systems, from PCs to workstations to supercomputers. ANSYS features file compatibility throughout the family of products and across all platforms. The multiphysics nature of ANSYS allows the same model to be used for a variety of coupled-field applications, such as thermal-structural, magneto-structural, and electrical-magnetic-flow-thermal. A model generated on a PC can also run on a supercomputer. This ensures enterprise-wide, flexible engineering solutions for all ANSYS users.

For both new and experienced users, the program offers a growing list of capabilities, including: advanced structural nonlinearities, electromagnetics, computational fluid dynamics, interactive design optimization, general contact surfaces, adaptive meshing, p-method adaptivity, large strain/finite rotation capability, and parametric modeling. The Motif-based menu system prompts data input and function selections through dialog boxes, pull-down menus, and sub-menus; helping users navigate through the program. Solid modeling features include NURBS-based geometry representation, geometric primitives, and Boolean operations (provided by the SHAPES[™] geometry engine from XOX Corp., which is incorporated into the ANSYS program).

ANSYS design analysis and optimization capabilities can be easily applied to CAD models through the use of IGES and STEP as geometry transfer tools, or through interfaces created with leading CAD programs. The ultimate goal is to integrate ANSYS technology with major CAD environments. Development work continues to provide similar integrated design analysis and optimization capabilities within other leading CAD packages.

User Interface

Although the ANSYS program has extensive and complex capabilities, its organization and user-friendly graphical user interface (GUI) make it easy to learn and easy to use. The program utilizes a comprehensive GUI based on the Motif Standard.

Through the GUI, the ANSYS program provides the user with easy, interactive access to program functions, commands, documentation, and reference material; providing a road map that teaches new users how to use the program by leading them stepby-step through an analysis. At the same time, the program offers full on-line documentation and a state-of-the-art, hypertext-based HELP system to assist experienced users in advanced applications. ANSYS developed an intuitive menu system to help the user navigate through the program. Users can input data through a mouse, a keyboard, or a combination of the two.

Within the user interface, there are four general methods for instructing the ANSYS program:

- Menus
- Dialog Boxes
- Toolbar
- Direct Input of Commands

Menus are groupings of related functions for operating the ANSYS program located in individual windows. The user can access these windows, which can be moved or hidden with a mouse, at any point in the process. ANSYS commands are mapped into functional groupings to provide quick access at appropriate points during an ANSYS session. The seven main menu or window areas, which are depicted in Figure 1, include:

- Utility Menu: Contains ANSYS utility functions that are mapped here for access at any time during an ANSYS session. These functions are executed through smooth, cascading pull-down menus that lead directly to an action or dialog box. The utility menu is modeless, so the user can complete more than one action at a time (e.g., changing view in the middle of a select operation).
- Main Menu: Comprises the primary ANSYS functions, which are organized in pop-up side menus based on the progression of the program.
- Input Window: Provides an input area for typing ANSYS commands, and displays program prompt messages. A command history is provided for previously typed commands. It allows for cutting and pasting commands from the log file, command history, and/or input files.

- Graphics Window: Represents the area for graphics displays, such as a model or graphically represented results of an analysis. The user can adjust the size of the graphics window, reducing or enlarging it to fit personal preferences.
- Output Window: Records the ANSYS response to commands and functions. This window is always accessible under the GUI.
- **Toolbar:** Permits the user to place commonly used functions, such as commands, or user-written routines for instant one-click access.

Utility Menu	
Toolbar	The state provide the set of the
Input Window	A CONTRACTOR OF A CONTRACTOR O
Main Menu	Reserved Barrier and Ba
Sub Menu	
Graphics Display	
Dialog Box	And
Output Window	

Figure 1

The use of the Motif Standard gives the ANSYS program a familiar look and feel, as well as the ability to access any control, function, or option from the same display.

• **Dialog Boxes** are windows that present the user with choices for completing operations or specifying settings. These boxes prompt the user to input data or make decisions for a particular function (Figure 2).

The **toolbar** represents a very efficient means for executing commands for the ANSYS program because of its wide range of configurability. It provides the user with the capability to create named buttons and have immediate access to commonly used commands (Figure 3). The toolbar can accommodate up to 200 buttons.

Regardless of how they are specified, commands are ultimately used to supply all data and control all program functions. The user interface is designed to make command selection and execution an easy, intuitive



Figure 2

Dialog boxes help the user navigate through the program by appearing whenever the user requires input for a particular function.

was ANSYS Toolbar	×
SAVE_DB ANSYSWEB	
RESUM_DB	
QUIT	
POWRGRPH	

Figure 3

The toolbar allows users to create buttons and have immediate access to commonly used commands.

process through the use of menus, dialog boxes, and the toolbar. The interactive nature of the user interface and the functional mapping of commands makes the actual command syntax transparent to the user. However, those users who are familiar with ANSYS commands can opt to input them directly via the keyboard.

Once executed, a command is listed in a session log file by the program. This log file is accessible through

the program's output window, permitting the user to review a list of commands in the event of an error, or to save a list of commands as a file for batch processing.

The program also makes use of progress bars that indicate the status of the operation during potentially long procedures (such as meshing). The user has the ability to stop the operation by a simple mouse click.

The complete ANSYS User's Manual Set and a hypertext-based HELP system are provided on-line to assist the user in operating the program to correctly complete an analysis. The user can retrieve detailed information on program functions, commands, and procedures; often through one or two mouse clicks. Users can get text, diagrams, and other program information by selecting a hypertext block in the main HELP index, or by using the system's word search capability. Users can just type in the topic for which they need information (e.g., nonlinearities), and the program will do the rest.

The ANSYS program also supports a host of graphics display options through X Windows, OPEN GL, and several other three dimensional (3D) graphic systems.

Graphics

Full interactive graphics are an integral part of the ANSYS program. Graphics are important for verifying preprocessing data and reviewing solution results in postprocessing.

ANSYS PowerGraphics gives the user significant speed when plotting ANSYS geometry and results. This speed is a result of storing the geometry as an object in memory rather than repeatedly assembling the data. PowerGraphics visualization features are available for element and contour displays, and are applicable to both p- and h-elements.

ANSYS PowerGraphics features allow for speed of Isosurface displays, Section/Capped/Q-Slice displays, and topological displays on Q-Slice. ANSYS graphics capabilities include the following:

- Boundary condition displays on solid models and finite element models
- Color contour displays of results
- Graphs of results vs. time or along a path in a model

- General display manipulation (viewing direction, zoom, magnification, rotation)
- Rubber-banding for solid primitives
- Multiple display windows
- · Hidden-line, section, and perspective displays
- Software Z buffering (smooth shading and faster display)
- · Light-source shaded images
- Edge displays (removal of interior element outlines for clarity)
- Shrink displays (separation of adjacent element lines from each other for clarity)
- Distorted ratio displays (independent scaling in horizontal and vertical directions for better visualization)
- Creation of composite displays of multiple entities (e.g. elements superimposed on the solid model)
- Up to 256 colors
- Three-dimensional (3D) volume visualization, including gradient display, isosurfaces, flow particle tracing, and volume slicing
- Presentation-quality X-Y data display, including multiple independent curves, 2D, and 3D displays; and control over colors, backgrounds, grid lines, and thicknesses
- Graphical progress status of plotting, meshing, listing, and solution
- Annotation capability to enhance graphics displays with text, dimensions, polygons, symbols, pie charts, etc.
- Animation support for animating deformed shapes, results over time, Q-Slicing, and Isosurfaces
- Color specification for most entities (including elements, lines, areas, volumes, boundary conditions, screen and outline colors, and color indices) by range or type
- Translucency specification for elements, solid modeling entities, component groups, and isosurfaces
- Pipes, elbows, beams, and magnetic sources shown in their actual shapes and cross-sections
- Display of composite material layers and orientations for layered elements

- · Color selection for window backgrounds
- Storage of display specifications on a file for later callback
- Hard copy graphics capabilities including Postscript, HPGL, and TIFF

Processors

ANSYS functions are organized into groups called processors. The ANSYS program has one preprocessor, one solution processor, two postprocessors, and several auxiliary processors such as the design optimizer. The ANSYS preprocessor allows the user to create a finite element model and to specify options needed for a subsequent solution. The solution processor is used to apply the loads and boundary conditions, and then determine the response of the model to them. With the ANSYS postprocessors, the user retrieves and examines the solution results to evaluate how the model responded, and to perform additional calculations of interest.

Database

The ANSYS program uses a single, centralized database for all model data and solution results (Figure 4). Model data (including solid model and finite element model geometry, materials, etc.) are written to the database using the preprocessor. Loads and solution results data are written using the solution processor. Postprocessing results data are written using the postprocessors. Data written to the database while using one processor are therefore available, as necessary, in the other processors. For example, the general postprocessor can read the solution data as well as the model data, and then use them for postprocessing calculations.

File Format

Files are used, when necessary, to pass data from one part of the program to another, to store the program database, and to store program output. These files include the database file, the results file, the graphics file, and so on. Files that are written by the program are in either ASCII (i.e., can be easily read or edited) or binary format. By default, the ANSYS program writes binary files using an external format (IEEE Standard),





which allows transportability across different hardware systems. For example, model geometry data can be created by one user on one computer system and then conveniently transferred to another ANSYS user on another system.

Reducing Design and Manufacturing Costs with ANSYS FEA

The ANSYS program allows engineers to construct computer models or transfer CAD models of structures, products, components, or systems; apply operating loads or other design performance conditions; and study physical responses, such as stress levels, temperature distributions, or the impact of electromagnetic fields. Engineers also use the program to optimize a design early in the design process, which reduces production costs. These processes help design engineering organizations abbreviate the multiple-prototype building, testing, and rebuilding cycle; as well as eliminate expensive product overdesigning.

In some environments, prototype testing is undesirable or impossible. The ANSYS program has been used in several cases of this type, including biomechanical applications such as hip replacement and intraocular lenses. Other representative applications range from heavy equipment components, to an integrated circuit chip, to the bit-holding system of a continuous coal-mining machine.

Engineers using ANSYS can pinpoint a potential design defect or determine an optimum design with the ANSYS design optimization feature before a design is in production or use. For example, an engineering consulting firm applied the ANSYS design optimization capability to an elastic disk used in the clutch assembly of an automobile engine. The goals were to extend fatigue life and to achieve a uniform stress distribution in the disk, while staying within geometric and mechanical interface constraints. Through its design optimization procedure, the program performed a series of solutions on a parametrically defined solid model of the disk, automatically adjusting selected dimensions at each solution until the optimum shape was achieved. Results showed that the difference between maximum and minimum von Mises stress within the disk was reduced by 27 percent, maximum stress was reduced by 28 percent, and fatigue life was increased by 35 percent. ANSYS design optimization enabled the engineers to reduce the number of costly prototypes, tailor rigidity and flexibility to meet objectives, and find the proper balance in geometric modifications.

Competitive companies look for ways to produce the highest quality product at the lowest cost. ANSYS FEA can help significantly by reducing design and manufacturing costs, and by giving engineers added confidence in the products they design. FEA is most effective when used at the conceptual design stage, as shown in Figure 5. It is also useful when used later in the manufacturing process to verify the final design before prototyping.

Program Availability

The ANSYS program operates on 486 and Pentiumbased PCs running Windows 95 or Windows NT, and workstations and supercomputers primarily running the UNIX operating system. ANSYS, Inc. continually works with new hardware platforms and operating systems. Contact your ANSYS Support Distributor for the most current list of hardware systems, configuration requirements, and pricing information.



Figure 5

FEA yields the greatest benefit when incorporated early in the design and manufacturing process.

ANSYS Family of Products

The ANSYS family of products consists of an extensive set of flexible, integrated multiphysics offerings that address enterprise-wide engineering requirements (Figure 6). ANSYS/Multiphysics[™], the company's flagship program, is the most comprehensive coupledfield, multiphysics software in the world. In addition to ANSYS/Multiphysics, the company's design analysis software programs are available as subsets, standalone programs, or enabled with add-on modules that enhance usability and functionality.

All of the capabilities of the ANSYS/Multiphysics program are available on a single compact disk. An authorization file, provided to customers with the license agreement, unlocks modules based on the sophistication of the user and the functionality they require.

Three categories describe the capabilities of the numerous offerings of the ANSYS family of products.

Multiphysics and Subset Offerings:

- ANSYS/Multiphysics, the company's most broad-based offering, is a coupled-field, multidisciplinary analysis program that enables users to study not only individual analysis capabilities such as structural, thermal, fluid flow, and electromagnetic, but the interaction of these analyses as well. This highly sophisticated design analysis software provides the design optimization capabilities necessary to simulate real-world engineering problems.
- ANSYS/Mechanical[™] provides a wide range of engineering design, analysis, and optimization capabilities including complete structural, thermal, and acoustics solutions. This design verification software is a powerful tool for determining displacements, stresses, forces, temperature, and pressure distributions; as well as other important design criteria.
- ANSYS/Structural[™] performs high-end structural analysis with advanced nonlinear capabilities including geometric, material, element, and buckling. This simulation tool enables users to accurately simulate the performance of large, complex models.
- ANSYS/LinearPlus[™], derived from the ANSYS/Mechanical product, is a low-cost structural analysis option designed for linear static (nonlinear gap contact and beam/shell large deflection), dynamic, and buckling analyses.
- ANSYS/Thermal[™] is a self-contained, standalone thermal program, also derived from the ANSYS/Mechanical product, for steady-state and transient thermal analyses.
- The ANSYS/PrepPost[™] program provides extensive, fully parametric features in the preprocessing stage, allowing users to quickly and easily create finite element models. The postprocessor enables users to examine results from all ANSYS analysis types.

 ANSYS/ED[™] is a fully functional design simulation program possessing the capabilities of the ANSYS/Multiphysics program with limits on problem size. This affordable, self-contained package is ideal for training and educational purposes.

Standalone Programs:

- ANSYS/FLOTRAN[™] offers flexible computational fluid dynamics (CFD) software with the capability of solving a variety of fluid flow problems, including laminar, turbulent, compressible, and incompressible flow. Coupled with ANSYS/Mechanical, ANSYS/FLOTRAN becomes the only CFD code with design optimization capabilities, providing sophisticated multiphysics capabilities.
- ANSYS/EMAG[™], a self-contained electromagnetic analysis package, simulates electromagnetic fields, electrostatics, circuits, and current conduction. This program, when used with other ANSYS products, generates multiphysics capabilities enabling the study of the interaction of flow, electromagnetic fields, and structural mechanics.
- ANSYS/LS-DYNA[™] is an explicit solution product option that solves highly nonlinear structural dynamic problems. This program, which simulates material forming, crash analyses, impact involving large deformation, nonlinear material behavior, and multibody contact, can be added to an existing license or used as a standalone product.

Designer Products:

- ANSYS/ProFEA[®] is a design analysis software with a streamlined version of ANSYS capabilities that allows users to analyze and optimize designs up-front within Pro/ENGINEER.
- ANSYS/AutoFEA[™] 3D, a seamlessly integrated design analysis program running completely inside AutoCAD release 13 and Autodesk Mechanical Desktop, lets engineers and designers assess the integrity of their designs during the conceptualization stage.

ANSYS Family of Products

		Tother,	hanica ,	Ictural	FULT -	earplus	T.mai	OTRAN	ag .	FER "	-oner	Cr. JSODD
	AND	ANS)	ANOTO NO	AWSY	ANSPO LIS	ANGP.	AWSP	AWSP	ANS, ST	AND CAL	AUCTO	AWSY'S
Linear Stress	*	*	*		*				*	*		*
Dynamic Analysis												
Modal	*	*	*		*				*	*		*
Spectrum	*	*	*		*							*
Harmonic	*	*	*		*							*
Random Vibration	*	*	*									*
Structural Transient												
Linear	*	*	*		*							*
Nonlinear	*	*	*	*								*
Structural Nonlinear												
Geometric	*	*	*	*	*							*
Material	*	*	*	*								*
Element	*	*	*	*	*							*
Buckling												
Linear	*	*	*		*							*
Nonlinear	*	*	*	*								*
Substructuring	*	*	*									*
Heat Transfer	*	*				*	*		*	*		*
Transient Thermal	*	*				*	*					*
Thermal Nonlinear	*	*				*						*
CFD	*						*					*
Electromagnetics	*							*				*
Electrostatics	*							*				*
Coupled-Field												
Acoustics	*	*										*
Piezoelectrics	*	*										*
Thermal/Structural	*	*										*
Elec/Mag/Therm/Structural	*											*
Solvers												
Frontal	*	*	*		*	*		*	*	*		*
Iterative	*	*	*		*	*	*	*	*			*
Explicit				*								
Preprocessing	*	*	*	*	*	*	*	*	*	*	*	*
Postprocessing	*	*	*	*	*	*	*	*	*	*	*	*
Optimization	*	*	*	*	*	*	*	*	*			*

The following optional modules can be added to the products specified.

	/Thermal	/LinearPlus	/FLOTRAN	/Emag	/LS-DYNA	/PrepPost
ANSYS/Multiphysics					*	*
ANSYS/Mechanical			*	*	*	*
ANSYS/Structural	*		*	*	*	*
ANSYS/LinearPlus	*		*	*		
ANSYS/Thermal		*	*	*		

Figure 6

ANSYS provides a robust range of geometry transfer/ import capabilities for simulating the performance of CAD models, including:

- STEP
- ACIS® (SAT)
- IGES
- Computervision/CADDS*
- Pro/ENGINEER[®]
- Unigraphics*

Converting legacy analysis data is simple with ANSYS. Translators are available for converting data from the following analysis codes:

- ABAQUS
- ALGOR
- COSMOS
- I-DEAS
- NASTRAN
- PATRAN
- STARDYNE
- WECAN

Example Used Throughout This Brochure

An example of a three-spoke pulley wheel mounted on a shaft is used in this brochure to help illustrate many ANSYS program capabilities and the flexibility they offer. A flat belt drives the pulley in a counterclockwise direction around the shaft (Figure 7).

Throughout this brochure, different loadings and different types of analyses are shown for the pulley assembly, depending on the capability illustrated. In each case, a two dimensional (2D) study has been done. Text that normally appears on the displays of this example has sometimes been removed for clearer viewing of the model.

A Note About This Brochure

The goal of this publication is to provide information regarding the ANSYS family of products. Much more information can be obtained from ANSYS documentation manuals. However, because of the breadth of the program, you may have additional questions that can best be answered by contacting your ASD or ANSYS, Inc.



Shown above is the finite element mesh of the pulley.

Preprocessing

An ANSYS analysis consists of three phases: preprocessing, solution, and postprocessing. In the preprocessing phase, data needed to perform a solution is specified. The user selects coordinate systems and element types, defines real constants and material properties, creates solid models and meshes them, manipulates nodes and elements, and defines coupling and constraint equations. The user can also employ a run statistics module to calculate the expected file sizes and memory requirements for solution.

Coordinate systems are used in the ANSYS program to locate geometry in space, to specify degree of freedom directions at nodes, to define material property directions, and to change graphics displays and listings. Cartesian, cylindrical, spherical, elliptical, and toroidal coordinate system types are available, all of which can be located anywhere in space and in any orientation.

Data entered by the user in the preprocessing phase becomes part of the centralized ANSYS database. This database is organized in tables of coordinate systems, element types, material properties, keypoints, nodes, loads, etc. Once data for a table is specified, it can be referred to by the table entry number. For example, several coordinate systems can be defined and later activated by simply referring to the appropriate coordinate system number (table entry number). In addition, a set of database control commands is available that allows portions of the database to be selected for specific operations. Users can select what they need based on criteria such as geometric locations, solid modeling entities, element types, material types, and node and element numbers, etc. For example, complicated boundary conditions can easily be specified or altered based on geometric location rather than on node or element numbers (Figures 8 and 9). The user may input a variety of information pertaining to the model but, during solution, the program will use only the portion of the database needed for the particular analysis.

Another convenient way to select model data is to divide the model into components (or layers), which are groups of geometric entities defined by the user for clarity or logical organization. Components may be displayed in different colors to clearly show various parts of a complex model.

The ANSYS program provides extensive model generation features that allow the user to quickly and easily create finite element models of actual engineering systems. In the ANSYS program, there are three different methods for generating a model: importing, solid modeling, and direct generation. Each method has its own unique features and advantages. The user can choose between these methods or employ a combination of them to create an analysis model.



Figure 8

Users can select portions of the database for various tasks such as applying boundary conditions or constructing different segments of the model. Here, database selection is used to separately display portions of the pulley model.

Solid Modeling

The ANSYS preprocessor's solid modeling capabilities allow the user to work directly with the geometry of the model without concern for the specific entities (nodes and elements) of the finite element model. To facilitate model generation, the program separates the definition of geometry and boundary conditions from the creation of the finite element mesh. The user first describes the geometry (and boundary conditions, if desired) of the solid model. When done interactively, easy verification of input data is possible. The program meshes the resulting model, which determines node locations and element connectivity.



Graphics displays that show the pulley nodes and some arbitrary boundary conditions are useful for model verification before an analysis is performed.

Two approaches to solid modeling are available in the ANSYS program: top-down and bottom-up. In top-down solid modeling, the user only specifies the highest order entities of a model. Commonly used solid modeling shapes (such as spheres or prisms), called geometric primitives, can be created with a single menu pick. For example, the user defines a volume primitive, and the program automatically defines associated areas, lines, and keypoints. In bottom-up solid modeling, the user builds the model from the lowest order entity on up. The user first defines keypoints, then associated lines, areas, and volumes, in that order. Bottomup and top-down modeling techniques can be freely combined in any model. Top-down modeling has many solid modeling capabilities. One capability is primitives. Primitives allow for the direct specification of geometric shapes. Shapes such as circles and rectangles in 2D and blocks, spheres, cones, and cylinders in 3D can be defined quickly and easily in the ANSYS program. Once geometry entities have been defined, whether by primitives, reading in IGES data, or from direct generation, users can perform Boolean operations on these geometric entities.

Whether employing the top-down or bottom-up approach, the user can use Boolean algebraic operations to combine sets of data and thereby "sculpt" a solid model. The ANSYS program has a full compliment of Boolean operations, such as add, subtract, intersect, divide, glue, and overlap. Boolean operations on line, area, and volume primitives can save considerable time and effort in building complex solid models. The ANSYS program makes tolerance adjustments automatically for Boolean operations, saving the user time. Additionally, the working plane, a user-defined coordinate system, may be used as a cutting tool.

Other solid modeling features provide the ability to drag, extrude, rotate, move, or copy solid model entities. Additional capabilities include circular arc construction, tangent constructions, volume and area generation by sweeping and revolving operations, automatic intersection calculation of lines and areas, and automatic fillet generation. Control over mesh density, calculations of mass property, and component groupings are also available.

The ANSYS program stores sets of interrelated lists describing the vertices, edges, faces, and volumes of the object being modeled. In ANSYS terminology, these are keypoints, lines, areas, and volumes, respectively.

In ANSYS solid modeling, all lines are represented as nonuniform rational B-splines (NURBS). A line is a portion of a spline bounded on both ends by a keypoint. An area is a portion of a surface completely bound by three or more lines. Volumes are portions of solids that are completely bound by four or more areas.

NURBS-based representation of solid model entities facilitates a surface construction technique known as skinning (also called lofting). Using this technique, the user can define a set of cross-sectional curves and instruct the ANSYS program to automatically generate a surface that fits through those curves. The skinning technique enables the user to quickly and easily model complex shapes that have a changing cross-section, such as a ship's hull or a car body. An example of skinning is shown in Figure 10.



Figure 10

The ANSYS solid modeler makes complicated shapes, like this vase, easy to model and mesh. This model was generated using skinning.

The ANSYS GUI provides powerful model generation tools such as mouse picking, rubber-banding, and working planes. With a mouse, the user can define or retrieve nodes, keypoints, lines, areas, and other entities by "picking" their locations at a point defined by the cursor's position on the display screen. Rubberbanding permits use of the mouse to expand or contract a solid primitive. With a working plane, the user can quickly and easily locate or select 2D model entities from a 3D model.

Meshing

Once the solid model is created, the finite element model (nodes and elements) may be generated with as little as one additional pick. The ANSYS program provides four types of meshing: mapped meshing, free meshing, extrusion meshing, and adaptive meshing.

Mapped meshing requires that the user decompose the geometry into simple pieces and select the appropriate element attributes and meshing controls such that the mesh consists of only quadrilateral or brick elements. For 2D models, free meshing can use all triangular elements or a combination of quadrilateral and triangular elements. In the Quad-dominant mesher, triangular elements are only used to better match geometrical features of the model, or to improve element quality in the mesh. For 3D models, all tetrahedral elements are used.

Free meshing uses a smart element sizing algorithm that sizes the elements according to the geometry (curvative and line lengths), as well as the proximity of the geometric features to each other. The resulting mesh is of high quality.

Extrusion meshing extrudes a 2D mesh into a 3D model composed of brick and/or wedge elements. Extrusion includes both sweeping and revolving operations.

Adaptive meshing is a process in which, after creating a solid model with boundary conditions, the user instructs the program to automatically generate a finite element mesh, perform the analysis, evaluate the mesh discretization error, and resize the mesh through a series of solutions, until the measured error drops below some user-defined value (or until a user-defined limit on the number of solutions is reached).

Adaptive meshing can be used for linear static structural or linear steady-state thermal analysis. The adaptive procedure can simultaneously respond to multiple loading conditions. In addition, the user may select regions of a model where the mesh discretization error is relatively unimportant and exclude such regions from the adaptive meshing operations. The user may also customize the adaptive meshing procedure to suit individual analysis needs.

The ANSYS program permits the user to modify a meshed solid model. For example, nodes and element attributes can be changed. For models containing repetitive features, the user can model and mesh a pattern region of the model and then generate copies of that meshed region. Once the solid model has been meshed, the ANSYS program automatically provides solid model cross-reference checking to ensure the validity of any modifications by the user to the meshed model. This cross-reference checking prevents the user from incorrectly deleting or otherwise contaminating solid model and finite element model data. For example, meshed keypoints, lines, areas, or volumes may not be deleted or moved unless the user explicitly instructs the program to override its automatic checking.

Robust meshers that can mesh very complex models directly, instead of meshing individual pieces and assembling them, include the 2D Tri mesher, the 2D Quad-dominant mesher, and the 3D Tetrahedron mesher. The Tri and Quad-dominant meshers allow for both exterior and interior mesh control and include a reliable mesh smoothing algorithm that reduces the number of meshing failures, and transition-mapped meshing capabilities, resulting in faster meshing. The savings in the number of elements result from the mesher's ability to efficiently transition from small to large elements. The Quad-dominant mesher also has a transition-mapped meshing capability that automatically places nicely shaped quad patterns into simple geometries.

The Tetrahedron mesher creates reliable, highquality meshes efficiently with as few elements as possible. It provides one-step meshing with extensive model checking before meshing, and a mesh improvement stage after meshing that creates a high-quality mesh. In the preparation stage, the facets checking feature ensures that the boundary triangular facets don't selfintersect and are correctly oriented, and notifies users before meshing when a mesh cannot be made or when problems are anticipated. The mesh improvement stage inserts, deletes, and moves elements and nodes improving overall mesh quality.

Parametric Definition

The ANSYS solid modeler and the ANSYS Parametric Design Language (APDL) (see ANSYS Parametric Design Language section, page 62) are closely linked.

Parameters can be used to define geometric dimensions (and other specifications) of a model,

allowing variations of those dimensions in subsequent analyses. Parameter definitions are automatically saved to the ANSYS log file, an ASCII file that contains all inputs entered during an ANSYS interactive session. The parameter values in this file can easily be changed, and the file can be read into ANSYS to recreate the model with the revised dimensions.

A parametric log file that uses the solid modeling capabilities of the ANSYS program for creation of an analysis model is particularly well-suited for use with design optimization. Dimensions of an object can be specified as parameters. As the shape of the object changes during optimization, the solid and finite element models also change. Boundary conditions are applied automatically to the new solid model without user intervention. Additionally, the user will find that making modifications to existing finite element models in standard, non-optimization analyses is simplified by using parameters and solid modeling.



Figure 11

ANSYS graphics allow for sophisticated manipulation of model data. For example, in the display above, the window on the left shows a ¹¼ s-symmetry bearing mount model, and the window on the right shows the inside of the model after a clipping plane was used to remove part of the model. Clipped displays may be used to show stress contours within a solid, 3D model. With the powerful ANSYS solid modeling capability, users can change the solid model and regenerate the finite element mesh in a shorter time and with less effort than would be required to alter the nodes and elements of a directly generated finite element model (Figure 11). However, if desired, the user may bypass the solid modeler and input the finite element model by direct definition of nodes, elements, and boundary data.

Direct Generation of Models

With the direct generation method, models can be defined in ANSYS preprocessing by specifying the location of every node and the size, shape, and connectivity of every element. Many commands are available that allow the user to conveniently copy, reflect, and scale a given pattern of nodes or elements.

Nodes are used to locate elements in space, and these elements define the connectivity of the model. Both can be generated in the most convenient manner without concern for solution efficiency.

Direct generation of nodes and elements is wellsuited to beam or piping models, and small models with regular geometry. For larger and more complex models, however, solid modeling is the recommended approach. The ANSYS program enables the user to easily switch back and forth between direct generation and solid modeling, using various techniques, as appropriate, to define different parts of the model.

Solution

The user obtains analysis results in the solution phase after a model is built in the preprocessing phase. In this part of an ANSYS analysis, the user specifies analysis type, analysis options, load data, and load step options, and then initiates the finite element solution.

The specified analysis type indicates to the program which governing equations should be used to solve the problem. The general categories of available analyses include structural, thermal, electromagnetic field, electric field, electrostatic, fluid, and coupled-field analyses. Each category includes several specific analysis types, such as static or dynamic analysis. The user can further define the analysis type by specifying analysis options. For example, they can specify one of several Newton-Raphson options to solve nonlinear equations.

Specified load data and constraints define the boundary conditions of the model. Load data includes degree of freedom constraints, point loads, surface loads, body loads, and inertia loads. Specific loads will vary with the analysis type (e.g., a point load can be a force for a structural analysis and a heat flow for a thermal analysis).

Each configuration of loads is called a load step, and an analysis may consist of one or more load steps. The load values of a given load step may be changed gradually from those of the previous load step (i.e., ramped), or they may be step-changed to the new values. The latter method would be used, for instance, to simulate shock loading during a transient analysis.

Load step options are used to set output controls, convergence controls, and general load controls for each load step. For example, the user can define the number of substeps to be used for each load step, or whether the values should be ramped over the load step.

Specified constraints can be used to limit the applicable degrees of freedom at selected nodes. For example, the rotational and translational degrees of freedom at nodes along a fixed edge can be constrained appropriately for a structural analysis. In addition to defining constraints during the solution phase, constraints also can be specified during the preprocessing phase on a solid model or a finite element model. Solid model degrees of freedom constraints are automatically transferred by the program to the finite element model upon initiation of the solution calculations.

Additional features of the solution phase allow the user to change material properties and element-specific data such as thickness, reactivate and deactivate elements (birth and death option), specify master degrees of freedom (MDOFs), and define gap conditions.

After specifications are completed for the appropriate solution phase criteria, the solution can be executed. The user can instruct the program to solve the governing equations and compute the results for the selected analysis type. This is the computationally intensive part of an ANSYS analysis and requires no user interaction. It requires the most computer time and the least user time.

The ANSYS program automatically reorders the elements and nodes to produce the most efficient solution times.

Equation Solver

All ANSYS analysis types are based on classical engineering concepts. Through proven numerical techniques, these concepts can be formulated into matrix equations that are suitable for analysis using the finite element method.

A mathematical model consisting of discrete regions (elements) connected at a finite number of points (nodes) represents the system to be analyzed. The primary unknowns in an analysis are the degrees of freedom for each node in the finite element model. Degrees of freedom may include displacements, rotations, temperatures, pressures, velocities, voltages, or magnetic potentials; and are defined by the elements attached to the node. Corresponding to the degrees of freedom, stiffness (or conductivity), mass, and damping (or specific heat); matrices are generated as appropriate for each element in the model. These matrices are then assembled to form sets of simultaneous equations that can be processed by the solver.

By default, the frontal solver is used to process these sets of equations. The frontal solver simultaneously assembles and solves an overall stiffness matrix made up of the individual element matrices. This procedure progressively moves through the model, element by element, introducing the equations corresponding to the particular element's degrees of freedom. At the same time, degrees of freedom are solved and deleted (using Gaussian elimination) from the matrix as soon as possible.

The ANSYS frontal solver incorporates Rank-n logic, which facilitates parallel processing, since degrees of freedom are solved for in groups, rather than individually. By varying the size of the degree of freedom group (the "n" in Rank-n), the ANSYS program can be tuned by each hardware vendor for optimum performance on their machines.

The degree of freedom set present in the assembled matrix at any given time is known as the wavefront, which expands and contracts as degrees of freedom are introduced to and deleted from the matrix. After the wavefront has passed through all the elements and the response of each degree of freedom has been computed, postprocessing can be used to display integrated results for the entire model.

As an alternative to the default frontal solver, the user can activate either of two iterative solvers, which provide faster solution times and utilize less computer resources in analyzing large models. In almost all analyses, the software is faced with solving a series of linear simultaneous equations. Direct solvers, such as the frontal solver, calculate exact solutions for a set of linear simultaneous equations, while iterative solvers iterate to approximate solutions.

The ANSYS program includes three iterative equation solvers: a highly efficient solver, known as the PowerSolver, which is a Preconditioned Conjugate Gradient (PCG) solver, the Jacobi Conjugate Gradient (JCG) solver, and the Incomplete Cholesky Conjugate Gradient (ICCG) solver. Having access to three different solvers enables ANSYS users to maximize productivity by choosing the most appropriate solver for a particular problem. The frontal solver is very efficient for small to moderate-sized problems, while an iterative solver is generally preferable for large, complex problems.

The PowerSolver represents a significant technological breakthrough because it transcends the limitations of prior iterative solvers. This solver is extremely reliable and accurate, with a preconditioner specialized for finite elements. It is the only iterative solver that can handle constraint equations and shell elements. The PowerSolver is a new level of technology that permits analysis of complex problems on desktop workstations, providing order of magnitude faster solution times and significant disk savings for large, complicated problems. The PowerSolver is applicable to both h- and p-element analyses and may also be used as an option in subspace iteration modal analysis. Additionally, for linear analyses with higher order planar or tetrahedron elements, a superfast option exists that cuts another factor of two in run times and disk space requirements.

The iterative solver can be used to provide more efficient solutions to field problems (including acoustic, heat transfer, and electromagnetic field problems) and other large analyses having symmetric, sparse, positive, and definite matrices.

An explicit solver, ANSYS/LS-DYNA, is also available. The explicit solver allows users to efficiently perform dynamic analyses including general, high-speed, largestrain, impact/contact problems; crash-worthiness simulation; failure analysis; and material forming processes including metal, glass, and plastic. ANSYS/LS-DYNA solves highly nonlinear structural problems. The explicit solution method is accomplished without the formation of a stiffness matrix and is ideally suited to problems of short duration involving contact, large deformations, and nonlinear materials. ANSYS/LS-DYNA consists of the combination of ANSYS pre and postprocessing, specifically customized for LS-DYNA3D, and solution of the problem by the LS-DYNA3D explicit solver from Livermore Software Technology Corporation (LSTC).

Structural Static Analysis

The structural static analysis capabilities in the ANSYS program are used to determine the displacements, stresses, strains, and forces that occur in a structure or component as a result of applied loads (Figures 12 and 13). Static analysis is appropriate for solving problems in which the time-dependent effects of inertia and damping do not significantly affect the structure's response. This analysis type can be used for many applications, such as determining the stress intensities in fillets of mechanical components or predicting the stresses in a structure resulting from a temperature distribution.

Most mechanical and structural engineers are familiar with this type of analysis and have probably





Deformation of the pulley under belt loading is calculated in a linear 2D static analysis. The broken lines represent the undisplaced shape.

solved numerous static problems using classical analysis methods or equations from engineering handbooks. The ANSYS program solves static analysis problems by applying numerical techniques to these same, traditional engineering concepts. The governing equation for static analysis in the ANSYS program is:

$[K]{u} = {F}$

where [K] is the structural stiffness matrix and {u} is the displacement vector.

The force vector, {F}, can include concentrated forces, thermal loads, pressures, and inertia loads. Inertia relief calculations, in which the ANSYS program determines the accelerations necessary to counterbalance the applied loads, can also be performed.

Static analysis in the ANSYS program can also include nonlinearities such as plasticity, creep, large deflection, large strain, and contact surfaces. A nonlinear static analysis is usually performed by applying the load gradually so that an accurate solution can be obtained. (A complete list of nonlinearities and the approach used to account for these factors is presented in the Structural Nonlinearities section, page 22.)



Figure 13

Equivalent (von Mises) stress contours of the pulley under belt loading are shown. The darkest portions of the stress contour in the hub and spokes indicate areas of high stress.

Structural Dynamic Analysis

Structural dynamic analysis is used to determine the effects of time-varying loads on a structure or component. Unlike static analysis, a dynamic analysis takes into account damping and inertia effects of time-varying loads. Examples of such loads are as follows:

- Alternating forces (rotating machinery)
- Suddenly applied forces (impact or explosion)
- Random forces (earthquake)
- Any other transient forces, such as moving loads on a bridge

All dynamic analysis types in the ANSYS program are based on the following general equation of motion for a finite element system:

$$[M]{\{\ddot{u}\}} + [C]{\{u\}} + [K]{\{u\}} = {F(t)}$$

where:

- [M] mass matrix
- [C] damping matrix
- [K] stiffness matrix
- {ü} nodal acceleration vector

- {u} nodal velocity vector
- {u} nodal displacement vector
- {F} load vector
- (t) time

Through this equation, the ANSYS program determines the values of the unknowns, {u}, which satisfy equilibrium at (t) every time, with inertia and damping effects included. The numerical integration with respect to time, when required, is accomplished through either Newmark direct integration or mode superposition.

The ANSYS program is capable of performing the following types of dynamic analyses: transient dynamic, modal, harmonic response, response spectrum, and random vibration.

Transient Dynamic

Transient dynamic analysis (also known as time-history analysis) is used to determine the dynamic response of a structure subjected to time-dependent loads. There are three methods available for obtaining a transient dynamic solution: full transient dynamic method, reduced method, and mode superposition. All three methods are based on the general equation of motion for dynamic analysis.

Of the three methods, full transient dynamic is the most general and the most powerful. This solution method uses the full mass [M], damping [C], and stiffness [K] matrices of the governing equation for dynamic analysis. Because of this, it has full nonlinear capability and may include plasticity, creep, large deflection, large strain, stress stiffening, and nonlinear elements (such as contact surfaces). In addition, any type of structural load may be used, including nodal forces and imposed displacements, element loads (such as pressures and temperatures), and inertia loads (such as gravity and rotational velocities and accelerations).

The full transient dynamic method uses a singlestep procedure to calculate displacements and stresses. The solution of the motion equation is based on the Newmark direct integration scheme in conjunction with the Newton-Raphson method (to account for nonlinear effects). An automatic time-stepping option is available for the full transient dynamic method. This option allows for a variable integration time step, achieving a balance between accuracy of solution and economy of computer resources.

Another feature of the full transient dynamic method is the capability to model the kinematic behavior of flexible structures. A combination of nodal couplings and specialized elements can be used to represent hinges, universal joints, rigid or elastic links, hydraulic cylinders, and other features found in many flexible mechanical systems.

For applications in which nonlinear effects are assumed to be negligible, the user may take advantage of the speed of either the reduced or mode superposition transient dynamic methods. Both of these methods assume linear behavior. Although general nonlinearities are not included, a special gap condition is available that can be used for impact problems. These two methods are useful for studying the overall behavior of a structure prior to performing a more extensive full transient dynamic analysis.

For the reduced transient dynamic analysis, the [M], [C], and [K] matrices of the governing equation are assumed to be linear. These matrices are condensed (through Guyan reduction) and expressed in terms of a set of dynamic, or master, degrees of freedom. The Newmark direct integration method is used to solve the equations of motion, and constant time steps are employed. Loading may include nodal forces, imposed displacements, and gravity.

The solution for a reduced analysis consists of a two-step procedure. The first step is to solve for nodal displacements at the master degrees of freedom. If strains, reaction forces, stresses, etc. are desired, the optional second step, an expansion pass, can be performed to expand the solution at desired time points to the full degrees of freedom set.

The mode superposition method is similar to the reduced method, in that it is a multistep, linear analysis requiring constant time steps. However, there are several differences. This solution sums individual mode responses from a modal analysis to calculate a structure's total response. It requires that the user perform a modal analysis prior to any other solution step. They can base the modal analysis on reduced matrices (through Guyan reduction), or full matrices (through subspace iteration).

Solution output for all three transient dynamic methods: full, reduced, or mode superposition, is in the form of nodal displacements, strains, stresses, forces, etc., as functions of time. Each of these items can be displayed, in graph form versus time or any other item, using the time-history postprocessor. The general postprocessor can be used to review the entire structure at any time point in the transient dynamic method (e.g., to produce displaced-shape displays and stress contour displays).



Figure 14

Displays, such as these stress contours of a wave propagating along a bar under impact, help analysts evaluate the results of a nonlinear transient dynamic analysis.

Determining which method of transient dynamic analysis is most appropriate will depend on the application involved and the needs of the user. For time-dependent effects on a nonlinear structure, such as impact of an automobile bumper, a full transient dynamic analysis may be required (Figure 14). If nonlinear effects are negligible (such as in the analysis of simple piping systems, machinery parts, gears, etc.), or if a preliminary analysis of a nonlinear structure is desired, the user may take advantage of the speed of a reduced or mode superposition analysis.

Modal

Modal analysis is used to extract the natural frequencies and mode shapes of a structure. Modal analysis is important as a precursor to any dynamic analysis because knowledge of the structure's fundamental modes and frequencies can help to characterize its dynamic response. The results of this analysis also help determine the number of modes or the integration time step to be used in transient dynamic analyses. Additionally, some transient solution procedures require the results of a modal analysis. The ANSYS program permits a prestressed modal analysis, as well as running a modal analysis following a large deflection analysis.

For modal analysis, the ANSYS program assumes free (unforced), damped, or undamped vibration, described by the following equation of motion:

$$[M]{\ddot{u}} + [C]{u} + [K]{u} = 0$$

The equation is recast as an eigenvalue problem. For undamped cases (which are most common for modal analysis) the damping term, $[C]{u}$, is ignored and the equation reduces to:

 $([K] - w^2[M]){\overline{u}} = 0$

where w^2 (the square of natural frequencies) represents the eigenvalues, and $\{\bar{u}\}$ (the mode shapes, which do not change with time) represents the eigenvectors. For damped cases, the equation of motion reduces to:

$$([K] + iw[C] - w^2[M]){\overline{u}} = 0$$

Five methods of eigenvalue extraction are available for modal analysis: reduced (Householder-Bisection-Inverse iteration), subspace iteration, Block Lanczos, unsymmetric, and damped. The reduced method uses reduced matrices, while the remaining methods use full matrices. The unsymmetric method is used when the stiffness and/or mass matrices are unsymmetric, such as



Figure 15

A modal analysis of the pulley is performed to determine the third, fourth, and ninth modes. The broken lines represent the undisplaced shape.

in acoustical fluid-structure interaction analyses. The damped method is for situations where damping cannot be ignored, such as rotor dynamic applications. Both the unsymmetric and damped methods are based on the Block Lanczos algorithm.

Modal analysis is useful for any application in which the natural frequencies of a structure are of interest (Figure 15). For example, a machine component should be designed to produce natural frequencies that will prevent the component from vibrating at one of its fundamental modes under operating conditions.

Harmonic Response

Harmonic response analysis is used to determine the steady-state response of a linear structure to a sinusoidallyvarying forcing function. This analysis type is useful for studying the effects of load conditions that vary harmonically with time, such as those experienced by the housings, mountings, and foundations of rotating machinery.

The governing equation for harmonic response analysis is a special case of the general equation of motion, in which the forcing function, $\{F(t)\}$, is a known function of time varying sinusoidally with a known amplitude, F₀, at a known frequency w (and phase angle, $\frac{1}{2}$: The displacements vary sinusoidally at the same frequency, w, but are not necessarily in phase with the forcing function. Loading can be in the form of nodal forces, imposed displacements, or element loads. The user can obtain the displacement solution at specified frequencies in terms of either amplitudes and phase angles or real and imaginary parts.

Three methods are available to do a harmonic response analysis: full, reduced, and mode superposition. In a full harmonic response analysis, the full [K], [M], and [C] matrices are used. The presence of full, potentially unsymmetric matrices provides for several enhanced analysis capabilities including acoustics, piezoelectrics, and rotor dynamics. Full harmonic response is useful for complex steady-state response problems, such as analyzing the stresses in a rotor bearing or determining the frequency response of an acoustic speaker. This analysis method requires only one solution step.

The reduced and mode superposition methods may be used for linear structures, which have symmetric matrices. Both of these methods offer time savings over the full method, which can be used for unsymmetric matrices. The reduced method employs Guyan reduction to condense the [K], [M], and [C] matrices; and involves a two-step solution procedure (a reduced

$${F(t)} = {F_0 (\cos (w t + f) + i \sin (w t + f))}$$

solution and an expansion pass). The mode superposition method, also a multistep procedure, requires a modal analysis prior to further solution steps.

Response Spectrum

A response spectrum analysis can be used to determine the response of a structure to shock loading conditions. This analysis type uses the results of a modal analysis along with a known spectrum to calculate maximum displacements and stresses that occur in the structure at each of its natural frequencies. A typical response spectrum application is seismic analysis, which is used to study the effects of earthquakes on structures such as piping systems, towers, and bridges.

The response spectrum data is supplied as a response-versus-frequency function. Four different types of response spectra are allowed: the displacement spectrum, the velocity spectrum, the acceleration spectrum, and the force spectrum. The user can specify a single response spectrum (or a series of spectra at different damping ratios) at a set of points in the model (single-point analysis), or different spectra at different points (multi-point analysis). Response spectra may be used for base or nodal excitation.

When users perform a response spectrum analysis, the program calculates structural displacements for each mode. An overall response may then be obtained by combining all modes by one of the following methods: Wilson-CQC, Ten-Percent, Double-Sum, Square-Root-of-Sum-of-Squares, or a user-defined method.

A Dynamic Design Analysis Method (DDAM) spectrum is also available for U.S. Navy shock analysis. This spectrum type is a customized single-point analysis, which uses appropriate equations and factors.

Random Vibration

Random vibration analysis is a type of spectrum analysis used to study the response of a structure to random excitations, such as those generated by jet or rocket engines.

The procedure for random vibration is similar to a response spectrum analysis in that the modal analysis results are used in the spectrum solution. However, the spectrum used is a power spectral density (PSD)-versus-frequency curve, which is a statistical measure of the energy of random excitation. The PSD can be input in terms of displacement, velocity, acceleration, pressure, or force. The user can specify a single PSD spectrum at a set of points in the model or different PSD spectra at different points. PSD spectra may be used for base or nodal excitation.

A normal (Gaussian) distribution of the PSD is assumed; the response calculated by the ANSYS program is also normally distributed. Therefore, it is possible to predict the probability that the actual response will exceed the calculated response.

There are three sets of solution quantities available, independent of the type of PSD used. They are the displacement solution (displacements, stresses, strains, and forces), the velocity solution (velocities, stress velocities, force velocities, etc.), and the acceleration solution (accelerations, stress accelerations, force accelerations, etc.). Any number of these solutions may be requested for a given analysis.

Random vibration analysis is especially useful to the aerospace industry, where components must be designed to withstand the effects of flight conditions. For example, instantaneous acceleration data obtained from a missile in flight can be converted to PSD data, which can then be used in a random vibration analysis to determine the response of the missile's components.

Structural Buckling Analysis

Buckling analysis is used to determine:

- 1) The load level at which a structure becomes unstable.
- 2) Whether or not a structure is stable at a particular load level.

This analysis type is important for determining the stability of any load-carrying structure, such as a bridge or tower. Two types of stability analyses are available in the ANSYS program: linear (eigenvalue) buckling and nonlinear buckling.

Linear Buckling

Linear, or eigenvalue, buckling accounts for stress stiffness (see the Structural Nonlinearities section, page 22, for more on stress stiffening) effects where compressive stresses tend to lessen a structure's ability to resist lateral loads. As the compressive stresses increase, the resistance to lateral forces decreases. At some load level, this negative stress stiffening overcomes the linear structural stiffness, causing the structure to buckle.

The ANSYS program uses an eigenvalue formulation to perform linear buckling analysis. This formulation determines the scaling factors (eigenvalues) for the stress stiffness matrix that offset the structural stiffness matrix. The governing equation for linear buckling is:

 $([K] - [1]){u} = 0$

where:

- [K] structural stiffness matrix
- [S] stress stiffness matrix
- 1 eigenvalues representing the scale factors
- {u} eigenvector representing the buckled shape

The point at which buckling occurs is called the bifurcation point, because of the two paths the force-deflection curve can take after reaching that point. After exceeding the bifurcation point, the structure will either buckle or continue to take on load in an unstable state (Figure 16).

It is important to realize that linear buckling cannot account for any nonlinearities or structural imperfections. These factors, if present in an actual structure (as they usually are), would cause the buckling load to be lower than the analysis results. However, linear buckling is very efficient and therefore requires relatively little computer time compared to a nonlinear buckling analysis. It is useful for studying the general behavior of a structure before performing a nonlinear stability analysis, or for academic engineering studies.

Nonlinear Buckling

To determine buckling loads more accurately, nonlinear buckling analysis should be used. Nonlinear buckling analysis is essentially an application of large deflection. The Structural Nonlinearities section, page 22 describes how the ANSYS program updates the orientation of a structure's elements in a large deflection analysis using a combined arc-length (Riks)/Newton-Raphson method.

The approach used in the incremental Newton-Raphson procedure is expressed as follows for any given equilibrium iteration:

$$[K]_{i-1} \{ Du \}_i = \{F\} - \{F^{el}\}_{i-1}$$

where:

[K] i-1	stiffness matrix from the	
	previous iteration	

- $\{ \mathbb{D}\, u \}_i \quad \mbox{ incremental displacement vector,} \\ \{ u \}_i = \{ u \}_i 1 + \{ \mathbb{D}\, u \}_i$
- {u}i displacement vector at the current iteration
- {F} applied force vector
- ${F^{el}}_{i-1}$ elastic force vector based on displacements for iteration (i-1)

The ANSYS program performs nonlinear buckling analysis by monitoring Du through the iterative process. Normally, in a large deflection analysis, the change in displacements between equilibrium iterations will decrease as the structure converges to a stable configuration. If the structure is loaded beyond its stability limit, however, Du will increase from iteration to iteration (that is, the solution diverges). The limit (buckling) load is the load level at which the solution begins to diverge.

The limit load derived in a nonlinear buckling analysis is usually lower than the bifurcation point determined in a linear buckling analysis, as illustrated in Figure 16. This difference occurs because nonlinear buckling can take into account the initial imperfections and nonlinearities (geometric and material) that exist in real structures.



Figure 16

This comparison of bifurcation point, or linear buckling to limit load buckling, indicates the unconservative nature of linear buckling.

A second application of nonlinear buckling analysis is in a snap-through analysis. Many types of structures will reach a second stable state after buckling if the load continues to increase. An example of such a structure is a shallow arch, pinned at each end, with a downward load applied at its apex. The arch will begin to deflect downward, as the force increases, until it reaches its buckling point and can no longer resist the applied load. It will then snap through, inverting its shape, and begin to resist the load once more. This second stable configuration can be determined by allowing the iterative process to continue at or above the limit load until the problem converges.

The arc-length method is employed in both limitload and snap-through nonlinear buckling analyses. When using the incremental Newton-Raphson method alone, the stiffness matrix may become singular, such as when the structure either collapses completely or "snaps through." The arc-length method causes the Newton-Raphson equilibrium iterations to converge along an arc to the equilibrium path, thereby allowing the analysis to follow the load-deflection curve (Figures 17 and 18).



Figure 17

Nonlinear buckling analyses, such as the snap-through behavior of a shallow arch under displacement loading, can be done with the large deflection capabilities of the ANSYS program.

Structural Nonlinearities

Structural nonlinearities cause the response of a structure or component to vary disproportionately with the applied forces. Realistically, all structures are nonlinear in nature but not always to a degree that the nonlinearities have a significant effect on an analysis. However, if the engineer determines that nonlinearities affect the behavior of a structure to the extent that they cannot be ignored, a nonlinear analysis is required.

The ANSYS program solves both static and transient nonlinear problems. The user executes a nonlinear static analysis by subdividing the load into a series of incremental load steps and, at each step, performing a succession of linear approximations to obtain equilibrium. Each linear approximation requires one pass through the equation solver (known as an equilibrium iteration). Similarly, nonlinear transient problems are broken into a succession of time-varying load increments, with equilibrium iterations at each step. However, the transient case can also include the integration over time of inertial effects.

In a nonlinear analysis, the structure's stiffness matrix and load vector may depend on the solution and are, therefore, unknown. To solve the problem, the ANSYS program uses an iterative procedure based on the Newton-Raphson method, in which a series of linear approximations converges to the actual nonlinear solution. For static nonlinear analysis, the arc-length method can be employed to control convergence, as illustrated in Figure 18.

By the Newton-Raphson method, the stiffness



Figure 18

The arc-length method is used in the ANSYS program to cause the Newton-Raphson equilibrium iterations to converge along an arc to the equilibrium path until nonlinear convergence is reached.

matrix and/or load vector can be updated with each iteration. The Newton-Raphson equation is as follows:

 $[K]_{i\text{ -1}} \{ \mathsf{D} u \}_i = \{ F^A \} \text{ - } \{ F^{NR} \}_{i\text{ -1}}$

where:

[K]i-1	tangent stiffness matrix based
	on the deformed geometry from
	the (i -1) iteration
{D u }i	incremental displacement
	vector, $\{Du\}_i = \{u\}_i - \{u\}_{i-1}$
{u}i	displacement vector at the
	current iteration
$\{F^A\}$	applied force vector

{F^{NR}}_{i-1} Newton-Raphson restoring load based on displacement for iteration (i-1)

Both the subdivision of the load and the maximum number of equilibrium iterations at each substep can be controlled by the user. Equilibrium iterations will continue until convergence is achieved or the maximum iteration limit is reached. For all types of nonlinearities, convergence checking can be based on the out-of-balance force, $({F^A} - {F^{NR}}_{i-1})$, and/or the displacement increment from one iteration to the next, $({Du}_i)$.

In many nonlinear static analyses, the loading must be applied in increments in order to obtain an accurate solution. The load is ramped starting from the initial load (usually zero) up to the final load value of interest. The ANSYS program features a time stepping capability that will automatically increment the load to obtain accurate and convergent solutions. The user only needs to specify the final load level and the minimum and maximum step size to be taken.

In nonlinear transient analyses, the dynamic equilibrium equations are solved through the Newmark time integration method. The transient analysis is divided into discrete time points. The difference between any two consecutive time points is called the integration time step, which affects the accuracy of the solution. The user specifies an initial integration time step based on the loading conditions, the natural frequencies of the structure, and other factors. The ANSYS program features an automatic time-stepping capability which, depending on the response frequency and the degree of nonlinearities, increases or decreases the integration time step. This minimizes the number of time steps required for the solution, yet maintains accuracy.

In addition to automatic time stepping and the arc-length method, the ANSYS program provides other convergence enhancement features such as prediction, bisection, line search, and adaptive descent. Prediction activates a linear predictor on the degree of freedom solution at the beginning of each substep, while bisection and adaptive descent cause a solution to back up and restart if the solution is detected to be off-track.

As an option, an explicit dynamic solver, ANSYS/LS-DYNA, may be used to efficiently solve highly nonlinear problems including dynamic contact-impact problems, such as crash and metal forming simulations, deep drawing, superplastic forming, extrusion, and rolling.

In both static and transient analyses, the ANSYS program can represent many different types of nonlinear effects. These nonlinearities may be classified into three categories: material, geometric, and element.

Material Nonlinearities

A material nonlinearity exists when stress is not proportional to strain. The ANSYS program simulates various types of nonlinear material behavior. Plasticity, multilinear elasticity, and hyperelasticity are characterized by a nonlinear stress-strain relationship. Viscoplasticity, creep, and viscoelasticity are behaviors in which strain may depend on other factors such as time, temperature, or stress. The Newton-Raphson method accounts for nonlinear material behaviors.

To fully account for plastic material behavior in an analysis, three important concepts must be considered: the yield criterion, the flow rule, and the hardening law. The yield criterion measures the 3D stress state by computing a single-valued equivalent stress that is compared against the yield strength to determine when the material will yield (Figure 19). The flow rule predicts the direction in which strain will occur. The hardening law, which is applicable to materials that strain harden, describes how the yield surface expands or changes as the material strains.

The ANSYS program can use one of three yield criteria to predict when yielding will begin: von Mises, a modified von Mises (Hill), and Drucker-Prager. The yield criterion is of the form:

Where s_{eq} is a scalar equivalent stress formed from the components of the stress tensor, and s_y is a reference stress. For rate-independent plasticity, no plastic flow can occur when f < 0 and yielding occurs at f = 0. For



Figure 19

Stress reversal in a bar under cyclic load, as seen here in this postprocessing graph of an element stress-strain history, can be modeled using the nonlinear material capabilities of the ANSYS program.

rate-dependent plasticity (viscoplasticity), the reference stress may be specified as a function of the rate of plastic straining.

As an example, the von Mises criterion for equivalent stress is as follows:

 $\frac{1}{2} \left[(s_1 - s_2)^2 + (s_2 - s_3)^2 + (s_3 - s_1)^2 \right]$ For the von Mises criterion, yielding begins when $s_{eq} = s_y$, the uniaxial yield strength.

Once it is established that the yield criterion is satisfied, the flow rule determines the direction and magnitude of plastic strain. The flow rule can be written as:

$${de^{pl}} = 1$$

where $\{d \in {}^{pl}\}\$ is the increment of plastic strain, Q (the plastic potential) is a scalar function of the components of stress that determines the direction of straining, and l(the consistency parameter) is the magnitude of straining. The flow rule is associative (that is, Q equals

the yield function) for all yield criteria in the ANSYS program, except for Drucker-Prager in which the flow rule can be associative or non-associative.

Hardening laws determine how a material yield surface is changed as it deforms plastically. In strain hardening materials, subsequent reloading will cause the material to yield again only if the load exceeds the previous stress level. Two kinds of hardening laws are represented in the ANSYS program: isotropic hardening and kinematic hardening. Isotropic hardening describes a yield surface that expands the same in all directions and implies that an increase in tensile yield strength due to hardening results in an equal increase in compressive yield strength. Kinematic hardening predicts an increase in tensile yield strength and produces a corresponding decrease in compressive yield strength. This is known as the Bauschinger effect.

A particular combination of yield criterion, flow rule, and hardening law describes a unique plasticity behavior. The ANSYS program models the following behaviors: classical bilinear kinematic hardening, multilinear kinematic hardening, bilinear isotropic hardening, multilinear isotropic hardening, anisotropic behavior, Drucker-Prager, and Anand. A userdefined option is also available.

- Classical Bilinear Kinematic Hardening describes general metallic materials that are considered to be bilinear; having one elastic and one plastic slope. This option is applicable to most common, initially isotropic, engineering metals in the small strain region. A modified von Mises yield criterion is used with an associative flow rule. Kinematic hardening accounts for the Bauschinger effect.
- Multilinear Kinematic Hardening also describes metallic materials, but is more applicable for materials having stress/strain curves with more than two slopes. This option uses the overlay or Besseling model to characterize complex multilinear behavior by combining simple stress/strain responses. A modified von Mises yield criterion is used with an associative flow rule. Kinematic hardening accounts for the Bauschinger effect.
- Bilinear Isotropic Hardening describes general

metallic materials that are considered to be bilinear. This option is applicable to isotropic materials, and is preferred over kinematic hardening at higher strains. The von Mises yield criterion is used with Prandtl-Reuss flow equations. The Bauschinger effect is neglected.

- Multilinear Isotropic Hardening describes general strain-hardening materials, especially in conjunction with large strain. The von Mises yield criterion is used; however, the Bauschinger effect is not modeled by this material behavior.
- Anisotropic Behavior describes materials that behave differently in tension and compression or that have different behaviors in different directions. By applying isotropic hardening to anisotropic material, this option can represent the effects of work hardening. A modified von Mises yield criterion is used with an associative flow rule.
- **Drucker-Prager** describes granular materials such as rock, concrete, or soil. This option uses von Mises yield criterion with dependence on hydrostatic stress to simulate the increase in yield strength that is produced by an increase in confinement pressure (hydrostatic stress). The flow rule can be associative or non-associative. No hardening is assumed.
- The Anand Model describes the behavior of metals at elevated temperature, although it may also apply at lower temperatures. This is an isotropic, rate-dependent, strain-hardening model with input through material parameters rather than in the form of stress-strain curves. The Anand model uses a von Mises yield criterion with an associative flow rule.
- User-Defined Models may also be incorporated to define virtually any nonlinear material behavior. The user-programmed FORTRAN subroutine is linked with the ANSYS program, and is accessed in a manner similar to the other plasticity options.

In addition to the plasticity behaviors previously described, the ANSYS program offers several other specialized material behaviors.

Multilinear elasticity is a conservative type of nonlinear stress/strain relationship, wherein all the strains are recovered after the load is removed. A modified von Mises criterion determines the point of change from linear to nonlinear behavior for multiaxial stress states.

Hyperelasticity represents the large strain behavior of very nearly incompressible and rubber-like materials. Elastic or rubber-like materials may be modeled using the Mooney-Rivlin model, which is used to characterize material properties for hyperelastic materials. The constants for this model can be determined automatically from stress-strain data related to a full suite of tests:

- Uniaxial tension
- · Equibiaxial tension
- Planar tension (shear)
- Uniaxial compression
- Equibiaxial compression
- Planar compression (shear)

A Blatz-Ko function is available for compressible foam-type, polyurethane rubber materials. In addition, a user-programmable feature allows for customized material functions.

Viscoplasticity is a combination of plasticity and creep. The primary applications are metal forming processes such as rolling and deep drawing that involve large plastic strains and displacements with small elastic strains. The plastic strains are typically very large (e.g., 50 percent or greater), requiring large strain theory. Viscoplastic material properties are represented in the viscoplastic elements by the Anand model, as previously described.

Creep, a time-dependent stress-strain relationship, is also represented in the ANSYS program. Creep accounts for additional nonlinear strain under a constant load or reduced stress under a constant displacement (stress relaxation). There are three stages of creep, as illustrated in Figure 20.

The ANSYS program has the capability of modeling the first two stages (primary and secondary). The tertiary stage is usually due to large geometric changes



Creep consists of three stages, shown here on this strain-versustime graph.

("necking down"), and is not analyzed since it implies impending failure.

Libraries of creep strain rate equations are built into the ANSYS program for primary, secondary, and irradiation-induced creep. In addition to the preprogrammed equations, user-defined functions for primary and secondary creep may be input by linking FORTRAN subroutines with the ANSYS program.

Viscoelasticity is an elastic time-dependent stressstrain relationship that characterizes viscously flowing materials such as heated glass. The material behavior is represented by a series of Maxwell models that allow for both shear modulus and bulk modulus relaxation with respect to time and temperature.

Additional material models are available with the explicit dynamic option, ANSYS/LS-DYNA, including strain-rate plasticity, crushable foam, and damage models for composite materials; as well as the standard plasticity and hyperelastic models available in ANSYS.

Geometric Nonlinearities

Geometric nonlinearities occur when the displacements of a structure significantly change its stiffness. The ANSYS program can account for these types of geometric nonlinear effects: large strain, large deflection, stress stiffening, and spin softening (Figure 21).



Figure 21

A nonlinear, large deflection analysis is required to determine this force-deflection curve of a two-strut shallow arch.

Large strain geometric nonlinearities account for the large localized deformations that may occur as a structure deforms. There are no assumptions on the magnitude of the strains or rotations that occur in the material. The program accounts for large strain by adjusting element shapes to reflect the changing geometry.

Large deflection represents a change in global structural stiffness resulting from a change in element spatial orientation as the structure deflects. The strains are assumed to be small, but the rotations are large. The program accounts for large deflection by updating the element orientations as the structure deflects. ANSYS large rotation and consistent tangent stiffness capabilities are available for beam and shell elements.

Another ANSYS capability used for large deflection analysis is the simulation of follower loads that always act normal to the structure's elements. Element pressures are used to describe such loads.

For large strain and large deflection, the stiffness is affected by the displacements. Therefore, an iterative solution is required to solve for changes in stiffness, and the Newton-Raphson procedure is employed. The arc-length method is available for static analyses for cases where buckling or snap-through is a possibility.

Stress stiffening (also known as geometric stiffening, initial stress stiffening, incremental stiffening, or differential stiffening) accounts for an increase or decrease in structural stiffness due to the stress state. Physically, it represents the coupling between the inplane and transverse deflections within a structure. This analysis option is valid for any structure, but is most appropriate for structures that are weak in bending resistance. Such structures might include pressurized membranes or turbine blades rotating at a high speed.

The ANSYS program uses the stress state of a structure to calculate a stiffness matrix, [S], which is added to the normal stiffness matrix, [K]. The resulting stiffness matrix is used to solve for the new displacements. Accordingly, the governing equation for a static analysis using stress stiffening is:

$$([K] + [S]){u} = {F}$$

This analysis option is solved in an iterative manner similar to the large deflection analysis. The ANSYS program also has the capability to include prestressing effects in otherwise linear problems; such as modal, linear transient dynamic, and harmonic analyses. This is accomplished by prestressing the model to be used in the linear analysis. The prestressed analysis can be used to simulate stiffening effects such as the tensile radial stresses that occur in a spinning turbine blade, and their effect on the blade's natural frequencies.

In rotating bodies, spin softening is another nonlinear effect that is often important. Whereas stress stiffening accounts for a change (usually an increase) in stiffness due to stress, spin softening models a decrease in stiffness due to the deflections of the body, such as a turbine blade, in the plane of rotation. Usually, stress stiffening and spin softening are used together in analyses of spinning bodies. Spin softening is modeled in the ANSYS static analysis by reducing the stiffness terms, K, in the plane of rotation by an amount equal to the square of the angular velocity, w, times the mass term, M, to obtain the adjusted stiffness, \bar{K} :

$$\overline{K} = K - w^2 M$$

Element Nonlinearities

Nonlinear elements are those elements that have their own nonlinear behavior, independent of other elements. This behavior is typically characterized by an abrupt change in stiffness due to a change in status (such as a contact surface element changing from open to closed). Element nonlinearities provide various capabilities that are not normally possible with global nonlinearities. The ANSYS element library includes the following nonlinear elements:

- General Contact Surface Elements: General surface-to-surface contact elements that can include significant sliding and transmission of loads between surfaces. Elastic or rigid coulomb friction may be specified between surfaces. The element may be closed and sliding, closed and sticking, or open.
- Interface Elements: Elements that represent point-to-point contact with limited sliding or point-to-ground contact with significant sliding. Surface friction may be included. The element may be closed and sliding, closed and sticking, or open.
- **Reinforced Solid Element**: A solid element that represents concrete, rock, or composites with up to three different sets of directional reinforcing material. The solid portion of the element is capable of crushing, cracking, deforming plastically, and creeping, while the reinforcing materials in the element can include plastic deformation and creep behavior.
- Nonlinear Damper: A longitudinal or torsional spring-damper that has a nonlinear damping response. This element's nonlinearity is a continuous function that is evaluated at each iteration.
- Nonlinear Spring: A varying stiffness rate spring that has a conservative or non-conservative response. The user specifies the element's force deflection curve with up to 40 linear segments.

- Tension-Only/Compression-Only Spar: A bilinear element used to represent a cable (tension-only) or a gap (compression-only). The element may be tensioned or slacked for the cable option or compressed or open for the gap option.
- Shell with Wrinkle Option: A membrane shell that collapses or wrinkles under compression. The element may be tensioned in both directions, collapsed in one direction, or collapsed in both directions.
- **Combination Element:** A single element that has combined mass, damping, gap, spring, and slider effects. This element has a lock-up option that prevents the gap from opening once the gap has closed, and a break-away option that prevents the interface from closing once it has opened.
- **Control Element:** A powerful element consisting of mass, damper, and slider effects. It is used to remotely control portions of a structure under predetermined conditions through binary (on-off) controls or controls defined by quadratic functions. It can represent mechanical snubbers, friction clutches, thermostats, relief valves, electrical switches, etc.

Surface-to-surface contact problems can be modeled using ANSYS general contact elements. The user defines a pair of contacting surfaces and then, with one additional command, instructs the ANSYS program to automatically generate contactor elements between the surfaces.

Some types of contact can be modeled using coupling or constraint equations. These are more general capabilities that can also allow the user to model distinctive features such as rigid regions, pinned structural joints, sliding symmetry boundaries, and other special inter-nodal connections. Using these techniques enables the user to link nodal degrees of freedom in ways that elements cannot.

Another nonlinear capability related to changing element status is the element birth and death option. This option allows the user to activate or deactivate the contribution of an element to the matrices during the solution phase. It can be used to simulate the addition and removal of material (e.g., excavation and fabrication), the interaction of moving parts (e.g., chain and sprocket interaction), or any other application in which an element's contribution to the solution depends on its location. The birth and death option is available for most ANSYS elements.

Static and Dynamic Kinematic Analysis

Kinematics is a branch of mechanics that deals with motion in the abstract, without reference to force or mass. Two types of kinematic motion can be described: rigid-body and flexible-body. Rigid-body kinematics assumes that the flexibility of moving structural members has a negligible effect on the solution. Flexible-body kinematics accounts for the local deformations that occur in a structure as it moves, making it a more realistic approach for real-world applications (Figure 22).

The ANSYS program can analyze large 3D motions of flexible bodies as part of its large deflection and finite (large) rotation analysis capabilities. These capabilities are used when cumulative effects of motion play a critical role.

The following features enable the ANSYS program to analyze a structure undergoing large motion:

- The Newton-Raphson solution method
- The ability of 2D and 3D structural elements to undergo large rotations
- A 3D element representing a revolute joint
- A 3D element representing a linear actuator A transient dynamic analysis is used to account for

inertial effects on the kinematic behavior of a structure. The Newmark time integration method is important in a dynamic analysis involving large deflections and dynamic effects. As a structural model undergoes a large movement through space over a period of time, a high degree of accuracy is required to solve for the model's dynamic response at each time point. The Newmark time integration method provides this accuracy, as it introduces little numerical damping.

ANSYS elements have been formulated to allow unlimited spatial motion in both 2D and 3D space. One example of this type of element is the 2D elastic beam element, a uniaxial element with tension-compression and bending capabilities. For kinematic studies, it has the ability to execute multiple planar rotations. This element might be used to model a type of crank linkage, such as an automotive windshield wiper, in which the rotary motion of one part of the structure (the crank) results in a reciprocating lateral movement in another part of the structure (the wiper). The 2D elastic beam element, with its infinite rotation capabilities, can be used to model the crank portion of the structure. This will allow the ANSYS program to track its movement and accurately determine the resulting motion occurring throughout the model.

The 3D revolute joint element represents a hinge or pin joint and is used to connect two parts of a model. The element is capable of representing a variety of effects, such as joint flexibility (or stiffness), friction, damping, and certain control features. However, the most important aspect of the revolute joint element is the capability of its axis to translate and rotate as the linkage moves.

The behavior of the joint's movement is determined by user-specified input. This includes specifications for the following:

- Friction torque
- Preload torque
- · Rotational viscous friction
- Interference rotation
- Joint flexibility
- Two differential rotation limits ("stops")
- Feedback control instructions

These input values determine the precise nature of the element's response. For example, the user can specify how the element is to behave if the upper or lower rotational stop is reached. The element can be directed to lock in place or bounce off the stop. Because the element has an independent coordinate system to track the pin axis movement, these values can remain unaffected by the joint's position relative to the global coordinate system. This allows the ANSYS program to accurately determine the relative position of other elements connected to the joint element.

The control features of the revolute joint element are provided by two of its five nodes. These nodes relay feedback control to the element according to degrees of freedom values selected by the user. As a result, the element can be made to change a particular aspect of its behavior according to some other change taking place elsewhere in the model. For example, the element friction torque value can be made to increase as the velocity of the component containing the element increases. Users can base such control decisions on the degree of freedom value itself, the first or second derivative of the value, the integral of the value with respect to time, or time itself. The control feature also allows any of the element's user-specified values, such as load or rotation, to be changed according to any of the decision criteria.



Figure 22

The ANSYS flexible body kinematics capabilities can analyze complicated, linked structures like the excavator shown here.

The linear actuator element models linkage members that rotate and experience a change in length. A hydraulic cylinder is an example of this behavior. This element is a tension-compression member with no bending stiffness. The ends of the element behave as pin joints. The user can specify an axial force or a displaced stroke length to change the length.

The revolute joint and linear actuator element, combined with the Newmark method and large rotation

capabilities, can be used to model a multiple-linkage, flexible-body mechanism for a large deflection analysis. Using these features, the ANSYS program can realistically evaluate the dynamics of complex motions through space and determine the resulting stresses, strains, and deflections that occur in the structure.

Thermal Analysis

The ANSYS program deals with the three basic methods of heat transfer: conduction, convection (both free and forced), and radiation. These types of heat transfer can be accounted for in a steady-state, transient, linear, or nonlinear thermal analysis.

The governing equation for heat transfer in a finite element system is:

 $[C]{T'} + [K]{T} = {Q}$

where:

- [C] specific heat matrix
- $\{T'\}$ time derivative of the nodal temperature
- [K] effective thermal conductivity matrix
- {T} nodal temperature vector
- {Q} effective nodal heat flow rate vector

Some of the thermal analysis capabilities available in the ANSYS program are:

- Steady-State
- Transient
- Phase Change
- Thermal-Structural

Steady-State

Steady-state thermal analysis predicts the equilibrium temperature distribution within a structure and the steady heat flow rates. Users can apply loads including convection surfaces, heat fluxes, heat flow rates, heat generation rates, and specified temperatures. The analysis may be linear or nonlinear.

In a linear steady-state heat transfer analysis, no thermal mass (specific heat) effects or temperature-dependent material properties are considered (Figure 23). The temperature derivative with respect to time, {T}, is zero and the effective thermal conductivity matrix, [K], is constant. The governing equation reduces to:

 $[K]{T} = {Q}$

This linear set of simultaneous equations is solved through a single iteration in the solution phase and is applicable to conduction and linear convection heat transfer.

In a nonlinear steady-state heat transfer analysis, time-dependent (thermal mass) effects are not considered. However, material properties (including convection film coefficients) may vary with changes in temperature, and radiation effects may be present.

Radiation can be defined three different ways in a thermal analysis. The radiation link element represents radiation between two points. The surface effect elements are useful for radiation between a surface and a point. A radiation matrix generator is available for problems involving several surfaces receiving and emitting radiation. This last option allows hidden or partially hidden surfaces, as well as a space node that can absorb or emit additional energy. In general, the heat flow rate, $\{Q\}$, for radiation is a function of T⁴ rather than T. It is a nonlinear process.

The conductivity matrix in a nonlinear analysis is expressed as a function of temperature and must be solved through an iterative procedure. In the ANSYS program, this procedure is based on the Newton-Raphson method, by which a series of linear matrix equations are successively solved to achieve a converged nonlinear solution. Accordingly, the equation for a nonlinear steady-state heat transfer analysis is:

$$[K]_{i} \{DT\}_{i+1} = \{Q^A\} - \{Q^{NR}\}_{i}$$

where i is the iteration number. The first iteration solves the equation at an assumed starting temperature (which may be specified by the user), and subsequent iterations use the temperatures from previous iterations to calculate the conductivity matrix. The iterative process continues until a converged solution is achieved; that is, when user-defined convergence criteria are met. Convergence checking can be based on the out-ofbalance load vector (heat flow) and/or the temperature increment from one iteration to the next.

For both linear and nonlinear steady-state heat transfer analyses, the solution data is in the form of nodal temperatures and heat flow rates. This data may be used in the postprocessing phase to produce displays of temperature contours (isotherms) through the model. Other postprocessing options may be used to extract more specific information, such as the thermal gradient and thermal flux at nodes and element centroids, and the heat flow rate across convection faces. Users can display this information in table or graph form.



Figure 23

The temperature increases caused by belt slippage on a jammed pulley are calculated in this linear 2D thermal analysis. The darkest portion of the temperature contour (upper right) indicates the region of maximum temperature.

Transient

A transient thermal analysis is used to determine the temperature distribution in a structure as a function of time, and to predict the rates of heat transfer and heat storage in a system. The transient analysis may be linear or nonlinear. The types of loads and nonlinearities that can be defined for a transient thermal analysis are the same as those discussed above for the steady-state case. Specific heat, which is input as a material property, is used to account for heat storage effects.

For transient thermal analyses, the governing equation (which includes a heat storage term) must be integrated with respect to time:

 $[C]{T'} + [K]{T} = {Q}$

This is accomplished through the Crank-Nicholson/Euler theta integration method in which the equation is solved at discrete time points within the transient. The difference between any two time points is known as the integration time step, which the user specifies. If necessary, the time step can be varied within the transient. The program's automatic time-stepping feature can be employed to automatically increase or decrease the integration time step based upon response conditions.

After the solution has been obtained, the postprocessing phase can be used to produce temperature contour displays and graphic or tabular output of more specific data (such as thermal gradient, heat flow, etc.) for any time point in the transient. Additionally, temperature-versus-time graphs and other data output can be obtained for specific points in the model.

Phase Change

A phase change analysis is a special case of transient thermal analysis that accounts for the solidification or melting of a material in the heat transfer process. This type of thermal analysis is useful in many applications, such as continuous metal casting processes or solar storage systems.

The energy released or absorbed when the phase change occurs (latent heat) must be accounted for in a phase change analysis. This is done in the ANSYS program by defining the enthalpy of the material as a function of temperature (Figure 24).

Since enthalpy is a relatively smooth function of temperature (compared to specific heat), convergence is enhanced. Also, the phase change cannot be missed by a time step that is too large, as can happen if specific heat is used to account for the latent heat.

In the ANSYS program, phase change analyses are solved through the same procedure used in any



Figure 24

In a phase change analysis, the ANSYS program accounts for the latent heat of phase change by using an enthalpy versus temperature curve.

other transient thermal analysis. The same types of results are also available for postprocessing. In addition, a "solid-liquid" contour display can be created by narrowing the displayed temperature range to that of the phase change region. Using the ANSYS animation capability, a series of these contours (at different times) can be displayed sequentially to visualize phase change propagation through the model.

Thermal-Structural

The thermal-structural analysis capabilities in the ANSYS program allow solution data from a thermal analysis to be input into a structural analysis. This feature is useful for determining the effects of temperature distributions on the structural response of the model. The user can apply the temperature load by itself or in conjunction with other mechanical loads.

Two methods of linking heat transfer and structural analysis are available in the ANSYS program. The first method involves performing two analyses in a series. A thermal analysis is first used to solve for the temperature distribution within the model from the given heat transfer boundary conditions. The temperatures from the thermal solution are then used as loads by the preprocessing and solution phases of a subsequent structural analysis of the model. The second method provides a simultaneous thermal-structural solution. This is made possible in the ANSYS program by using coupled-field elements that have both temperature and displacement degrees of freedom. The user constructs the analysis model using these elements and specifies thermal and structural boundary conditions. In the solution phase, each iteration calculates both the thermal solution and the structural solution based on the temperature and displacement data from the previous iteration. General contact elements may also be used in a coupled-field analysis. These elements allow for heat transfer across a contact interface. As two surfaces come into physical contact, they also begin transferring heat.

With the simultaneous solution process, it is possible to couple complex heat transfer and structural problems, such as transient thermal and nonlinear dynamic analyses. For example, this method may be used to analyze a bimetallic strip which, when heated, experiences coupled thermal and structural deformations. In such a situation, large geometric deformations may occur due to the different rates of expansion of the two metals, which may affect the conductivity matrix.

Electromagnetic Field Analysis

The electromagnetic capabilities in the ANSYS program, available as a stand-alone product (ANSYS/Emag), or in the ANSYS/Multiphysics product, can be used to analyze the different aspects of electromagnetic fields, such as inductance, flux density, flux lines, forces, power loss, and other related phenomena. These capabilities are effective for analyzing devices such as solenoids, actuators, motors, permanent magnet devices, transformers, and similar components.

Two classes of electromagnetic analyses can be performed:

- Analysis of 2D planar, axisymmetric, and 3D static electromagnetic fields
- Analysis of 2D planar, axisymmetric, and 3D low frequency time-varying electromagnetic fields

The finite element formulations used in ANSYS electromagnetic analyses are derived from Maxwell's equations for electromagnetic fields. By introducing a scalar potential or vector potential into Maxwell's equations and considering their constitutive relationships, users can develop equations that are suitable for finite element analysis.

Several other ANSYS features add power and flexibility to the program's electromagnetic capabilities. For example, the user can conveniently choose units for electromagnetic analysis as either CGS or MKS, or otherwise. As an alternative to the standard ANSYS frontal solver; the PCG, ICCG, and JCG iterative solvers can be very useful for electromagnetic field problems because they provide faster solutions to potential field problems. The 2D and 3D infinite boundary elements eliminate the need to model large extents of the infinite medium surrounding the electromagnetic device (e.g., air), resulting in smaller models and less demand on computer resources. Virtual work and Maxwell stress tensor force calculations are available for all electromagnetic elements.

One of the major advantages of using the full, multipurpose ANSYS program for electromagnetic FEA is its capability for coupled-field analysis. The coupled-field loads of an electromagnetic analysis (i.e., forces and heat generation) can be automatically coupled to ANSYS structural, thermal, and fluid elements. In addition, an electric circuit may be directly coupled with conductors or sources in an electromagnetic analysis to model circuit-coupled devices.

Static Electromagnetic Fields

Static electromagnetic field analysis can be performed in two or three dimensions for linear and nonlinear analyses. The finite element formulation for static analysis is:

$$\begin{split} & [K] \{ D U \}_k = \{ R \} - \{ F \} \\ & \{ U \}_{k+1} = \{ U \}_k + \{ D U \} \end{split}$$

where:

- [K] coefficient matrix
- {U} nodal potential vector
- $\{DU\}$ incremental nodal potential vector
{F} residual load vector

k iteration number

Two-dimensional magnetostatic field problems are solved by minimizing a nonlinear magnetic energy functional containing a vector potential ($U = A_z$), resulting in a set of simultaneous equations. Users obtain the solution in the ANSYS program through an iterative procedure based on the Newton-Raphson method. This solution method is available for the 2D coupled-field solid element, which also has structural and coupledfield capabilities. It is also available for the 2D higher order magnetic solid element. Current conductors and permanent magnets can be modeled as sources. Current sources may be fed by a known voltage or current. Other items that can be modeled include saturable irons, non-magnetic materials, and velocity effects.

Three-dimensional magnetostatic fields are solved by minimizing a nonlinear energy functional associated with a single scalar potential (U =, or a three-component vector potential (U = Ax, Ay, Az). The Newton-Raphson iterative procedure is used to solve 3D static electromagnetic problems. Current conductors and permanent magnets can be modeled as sources.

Conductors can be modeled with elements or described by bar, arc, or coil primitives; or electromagnetic field coupling. Users can also model saturable irons, non-magnetic materials, and velocity effects.

The ANSYS program provides for a variety of linear and nonlinear magnetic material representations including isotropic or orthotropic linear permeability, material B-H curves, and permanent magnet demagnetization curves. Postprocessing functions allow the user to display flux lines, flux density, and field intensity; and to perform calculations for force, torque, source input energy, inductance, terminal voltage, and other parameters (Figure 25).

Time-Varying Electromagnetic Fields

Time-varying electromagnetic field analyses can be performed for 2D planar and axisymmetric, or 3D problems. Two kinds of time-varying analyses are available in the ANSYS program: alternating current (AC) and



Figure 25

The ANSYS program is used to calculate magnetic flux lines in a typical induction motor.

transient. AC, or harmonic electromagnetic field analysis, solves for complex vector potentials (A) and scalar (# potentials, flux density (B), and field intensity; while transient electromagnetic field analysis solves for timevarying (real) vector potentials, flux density, and field intensity. Time-varying analysis is used to calculate the effects of eddy currents in a system. Making use of the coupled-field capabilities can allow for thermal, structural, and electric circuit coupling.

The finite element formulation for AC electromagnetic field analysis is:

```
[K + jw C]{A} = {F}
where:
[K] coefficient matrix
j --1
w angular frequency
[C] magnetic "damping" matrix
{A} nodal potential vector
```

{F} applied load vector (current or voltage loads)

The solution for the AC analysis can be expressed in terms of the complex potential (real and imaginary, or amplitude and phase angle) calculated at each node. Material property input for magnetic permeability and electrical resistivity may be constant or temperaturedependent. Postprocessing functions produce calculations for source impedance, power loss, eddy currents, stored electromagnetic energy, inductance, resistance, electromagnetic forces, and other field effects.

The finite element formulation for transient electromagnetic field analysis is:

 $[C]{A'} + [K]{A} = {F}$

where:

- [C] magnetic damping matrix
- {A'} time derivative of the nodal potential vector
- [K] coefficient matrix
- {A} nodal potential vector
- {F} applied load vector (current, voltage, or permanent magnet loads)

The Crank-Nicholson implicit time integration scheme, in conjunction with the Newton-Raphson method, solves this formulation in the time domain. The Crank-Nicholson integration scheme is a stepping procedure that calculates the vector potential field at discrete time points. The Newton-Raphson method is used at each discrete time point to resolve the material nonlinearities. Nonlinear B-H curves are allowed, and convergence in the nonlinear solution is achieved at each time point. As with any nonlinear analysis, electromagnetic or otherwise, the ANSYS program provides automatic time stepping. This feature allows the user to specify only the smallest time step and then instruct the program to automatically calculate the remaining time steps. Postprocessing commands, macros, and functions calculate power loss, eddy current density, and forces, either for each element or for the total system, at any point in the transient analysis.

Electric Field Analysis

ANSYS electric field analysis capabilities cover three areas of electric fields: current conduction, electrostatic analysis, and electric circuit analysis. Typical quantities of interest include current density, electric field strength, voltage distribution, electric flux density, Joule heat, stored energy, forces, capacitance, current, and voltage drop.

Electric field analyses can be conducted in 2D or 3D, and are useful in the design of many engineering components, such as bus bars, fuses, transmission lines, HV insulators, microstrips, shielding, capacitors, controllers, and circuits.

The program uses Laplace's equation as the basis for static electric field analysis. The program solves circuit problems using a nodal analysis method of the electric circuit theory. Primary unknowns (nodal degrees of freedom) calculated by the finite element solution are electric potentials (voltages). Other electric field quantities are then derived from the nodal potentials.

Electric Current Conduction

The ANSYS program can be used to conduct a steadystate current conduction analysis to determine the current density and electric potential (voltage) distribution due to direct current (DC) or potential drop. Two types of loads can be applied in this analysis: applied voltage and electric current. A steady-state current conduction analysis is assumed to be linear (i.e., the electric current is proportional to the applied current).

The finite element formulation for steady-state current conduction analysis is:

 $[K]{V} = {I}$

where:

- [K] coefficient matrix
- {V} nodal electric potentials
- {I} applied load vector (current)

The solution for a steady-state current conduction analysis uses potential functions and is expressed in terms of the nodal electric potentials (current density or voltage). In most cases, an electric current conduction analysis is followed by, or coupled with, a thermal analysis to calculate the temperature distribution due to Joule heat, or an electromagnetic field analysis to calculate the magnetic field produced by electric current (see the Coupled-Field Analysis section, page 41).

Electrostatics

An electrostatic analysis is used to determine the electric field and electric scalar potential (voltage) distribution due to charge distributions or potential drop. Two types of loads can be applied in this analysis: applied voltage and charge densities. An electrostatic analysis is assumed to be linear (i.e, the electric field is proportional to the applied voltage).

The finite element formulation for an electrostatic analysis is:

 $[K]{V} = {Q}$

where:

[K] coefficient matrix

{V} nodal electric potentials

{Q} applied load vector (charge)

The solution for an electrostatic analysis consists of nodal electric potentials from which electric field strength and flux density are calculated. Also calculated are forces arising from the electrostatic field via a Maxwell stress tensor approach. These forces can be applied directly by the program into a structural analysis.

Electric Circuit Analysis

Electric circuit analysis determines the voltage and current distribution in an electrical circuit due to applied source voltages or currents. The sources may be DC, AC, or time-varying. The electric circuit capability models linear circuit elements and includes the following circuit components: resistor, capacitor, inductor, mutual inductor, voltage-controlled current source, voltage-controlled voltage source, current-controlled current source, current-controlled voltage source, independent voltage source, and an independent current source. In addition, three other current sources can hook directly into an electromagnetic field model so that stranded conductors and massive conductors in the field model can be directly linked to an external circuit.

The finite element implementation is based on Kirchhoff's Current Law, using a simple lumped circuit approach. The overall formulation can be expressed in matrix terms as:

$$[C] \{V'\} + [K] \{V\} = \{i\}$$

where:

[K] coefficient matrix

[C] damping matrix

- {V} nodal electric potential vector
- $\{V'\}$ time derivative of the nodal electric potential vector
- {i} current vector

The solution for a static, harmonic, or transient analysis is a set of nodal voltages from which derived quantities are calculated for each circuit element (such as current and power).

The electric circuit elements may be coupled to an electromagnetic field model to simulate voltage-fed, or circuit-coupled, stranded and massive conductors. This capability allows for simulating devices controlled by external circuit connections such as solenoid actuators, transformers, electric machines, etc. When stranded or massive conductors are connected to an electric circuit, both electric current and voltage are unknown and must be solved for simultaneously. A modified nodal analysis method builds circuit equations for the coupled-field behavior.

Circuit-coupling is available for static, harmonic, and transient analysis for both 2D and 3D analysis. The circuit element may couple directly with the electromagnetic field elements.

For circuit-coupled stranded coils, the matrix equation is:

 $\begin{bmatrix} 0 & 0 & 0 \\ C^{iA} & 0 & 0 \\ 0 & 0 & 0 \end{bmatrix} \begin{pmatrix} A' \\ 0 \\ 0 \end{pmatrix} + \begin{bmatrix} K^{AA} & K^{Ai} & 0 \\ 0 & K^{ii} & K^{ie} \\ 0 & 0 & 0 \end{bmatrix} \begin{pmatrix} A \\ i \\ e \end{bmatrix} = \begin{pmatrix} 0 \\ 0 \\ 0 \end{bmatrix}$

For circuit-coupled massive conductors, the matrix equation is:

$$\begin{bmatrix} C^{AA} & 0 & 0 \\ 0 & 0 & 0 \\ C^{VA} & 0 & 0 \end{bmatrix} \begin{pmatrix} A' \\ 0 \\ 0 \end{pmatrix} + \begin{bmatrix} K^{AA} & 0 & K^{AV} \\ 0 & 0 & 0 \\ 0 & K^{iV} & K^{VV} \end{bmatrix} \begin{pmatrix} A \\ i \\ V \end{pmatrix} = \begin{pmatrix} 0 \\ 0 \\ 0 \end{pmatrix}$$

where:

- K^{AA} magnetic potential stiffness matrix
- Kⁱⁱ resistive stiffness matrix
- K^{Ai} potential-current coupling stiffness matrix
- K^{ie} current-emf coupling stiffness matrix
- C^{iA} inductive damping matrix
- C^{AA} potential damping matrix
- CVA voltage-potential damping matrix
- A nodal magnetic potential vector
- A' time derivative of the nodal magnetic potential vector
- i nodal electric current vector
- e nodal electromotive force drop
- V nodal electric voltage vector

The solution to a circuit-coupled electromagnetic field analysis is the circuit nodal potentials and the electromagnetic nodal potentials from which derived quantities are calculated including current in the circuit, resistance, inductance, magnetic field quantities B and H, Joule heating losses, eddy currents, forces, etc.

Fluid Flow Analysis

The ANSYS fluid elements enable the user to employ computational fluid dynamics (CFD) techniques such as sequential coupled analyses, or a standard pipe flow analysis to study the flow, pressure, or temperature distribution of a liquid or gas within a given system. The user can analyze transient and steady-state problems. Up to six nonreacting species may constitute the fluid.

Graphical solution monitoring enables ANSYS/FLOTRAN users to graphically track solution features for nonlinear steady-state solutions and transient solutions. The user receives continuous feedback on solution progress via a series of X/Y-type graphs that ANSYS displays and updates.

Computational Fluid Dynamics

CFD capabilities are provided through the integration of the ANSYS/FLOTRAN program, which is available as a stand-alone product or in the ANSYS/Multiphysics product. This robust capability is available for both 2D and 3D analyses through the integration of the two FLOTRAN elements within the ANSYS program, providing a powerful engineering tool for solving fluid flow and heat transfer design problems.

CFD analysis is used to determine flow characteristics of a fluid medium, such as pressure drop, velocity distribution, direction of flow, lift and drag forces, and heating or cooling effects. It can be used to solve for flow, pressure, and temperature distributions in a single-phase, viscous fluid. The fluid may be either Newtonian or non-Newtonian.

The velocity components, pressure and temperature, are calculated from the conservation of mass, momentum, and energy (Figure 26). A two-equation turbulence model is available for simulating turbulent flows. Derivative results include Mach number, pressure coefficient, total pressure, and stream function for fluid analyses; and heat flux and heat transfer (film) coefficient for thermal/fluid analyses.

Several types of CFD analyses are available, including:

- Laminar Flow: This is suitable for analyses in which the velocity field is very ordered and smooth, such as with highly viscous, slow moving flows. A laminar flow is considered to be incompressible if density is constant or if little energy is expended by the fluid in compressing the flow.
- **Turbulent Flow:** This is suitable for analyses in which the velocities are high enough and the viscosity is low enough to cause rapid fluctuations in the velocities are considered turbulent. The effect of the rapid fluctuations on the bulk fluid motion is handled throughout the turbulence model. Turbulent conditions at the wall are handled automatically. The flow is modeled as incompressible if density is constant, or nearly constant; or if little energy is expended by the fluid in compressing the flow.

• Thermal/Fluid: Users can calculate the temperature distribution throughout the flow field. In a "conjugate heat transfer problem", the energy equation is solved in a domain with both fluid and solid regions. In a natural convection analysis, the flow is generated from the pressure differentials brought about by density gradients caused by temperature variations.

In forced convection, there are externally applied flow forces. Conduction occurs in the fluid layer adjacent to a surface and then the fluid motion carries the energy away. Boundary conditions for thermal analyses include temperature, film coefficient, heat flux, and radiation.

• **Compressible Flow:** This flow regime is typically required for high-speed gas flows, where density changes significantly influence the nature of the flow field. The fluid velocities are significant compared to the speed of sound. Subsonic, transonic, or supersonic flows may be analyzed with or without heat transfer.

The finite element formulation for a fully-coupled flow equation system is as follows:

Kxx	Kxy	Kxz	-Cx	0		V I	=	E
Kyx	Kyy	Kyz	-Cy	0		V	=	F
Kzx	Kzy	Kzz	-Cz	0	4	V	י = נ	Ε
$C\mathbf{x}^{\mathrm{T}}$	$C_{\boldsymbol{y}^T}$	$C_{z^{\mathrm{T}}}$	0	0		P	=)
Ктх	\mathbf{K}_{TY}	Ktz	0	Kτ		Γ	=	F

The vectors V_x, V_y, V_z, P, and T represent five primary degrees of freedom; and contain the unknown nodal velocities, pressures, and temperatures; respectively. In the global coefficient matrix, the K sub-matrices represent coupling terms resulting from advection transport and diffusion. The C matrices are the pressure gradient operators and their transposes, and the C^{T} matrices are the velocity divergence operators. Finally, the right-handside vectors, (F), contain surface-flux type contributions, body forces, and in the case of transient flows, history effects from previous time levels.

In a fully-coupled formulation, the global equation system is solved for all the nodal unknowns in a simultaneous manner. If a turbulence model is also employed, the simulation may involve the additional primary variables of turbulent kinetic energy k, and its rate of viscous dissipation, e. As the size and physical complexity of the flow problem increases, the cost of the above solution technique becomes prohibitively expensive, both in terms of storage and CPU time. For this reason, ANSYS/FLOTRAN utilizes a segregated solution technique, where separate equation systems are assembled and solved for each of the primary flow variables in sequence. This minimizes the size of the matrix equation being solved at any time.

ANSYS/FLOTRAN provides three distinct solvers that can be used for CFD analyses. A Preconditioned Conjugate Gradient solver is used to solve the pressure equation for incompressible CFD problems. The Conjugate Residual solver, with or without preconditioning, provides solutions for nonsymmetric systems such as the energy equation, the pressure equation for compressible flow, or the multiple-species transport equations. The Tri-Diagonal Matrix Algorithm (TDMA) can be used to efficiently approximate solutions for any of the equation sets.

Some typical applications for CFD analyses include an evaluation of the lift and drag on an airfoil, the flow in supersonic nozzles, the complex 3D flow pattern in a pipe bend, and gas pressure and temperature distribution within an engine exhaust manifold as well as temperature distribution within the manifold itself. Natural or forced convection cooling of electronic components also can be analyzed.

The flow characteristics of fluids such as blood and various plastics can be simulated utilizing the non-Newtonian flow capability. Viscosity can be calculated with the Bingham, Carreau, or Power Law models. The user-defined viscosity model can be supplied also.

CFD elements provide the capability to do sequential CFD/structural and CFD/thermal analysis. A CFD analysis will show the convective behavior of a fluid medium. A more detailed thermal analysis, using CFD results as boundary conditions, could then be done on a structural model.

ANSYS users can couple magnetic Lorentz forces and Joule heating with their fluid flow analysis terms.



Boolean Operations

The ANSYS solid modeler provides Boolean algebraic operations (such as intersection, subtraction, and union), which can be used to "sculpt" a solid model. In this display, the hexagonal and block primitives are subtracted from the cylinder primitive to form the basic socket volume.

P-Elements

This image shows stress results for a pillow block bearing housing that was solved using the p-method. User-definable convergence checking allows areas that are not important for the stress raiser to be excluded.

Skinning

Skinning, also known as lofting, is a surface construction technique that allows the user to define a set of two or more lines and then instruct the program to automatically generate an area that fits through those lines, as in this solid model of a vase. The window on the left shows selected lines and four cross-sectional areas of the model that were defined before skinning the curved surface areas of the vase. The window on the right shows the four surface areas generated by skinning sets of lines on the cross-sections.



Examples

of ANSYS

Capabilities



3D Postprocessing

This model represents one-half of the bulkhead between cylinders one and two of an inline, six-cylinder engine block. In search of "lighter" engines, designers from Cummins Engine Co. Inc. used "thinner" sections and smaller fillet radii. By using the combination of Pro/Mesh and the ANSYS fast solvers, Cummins was able to determine that there would be a potential cracking problem, analyze several variations of the geometry, and resolve the issue in time to incorporate the necessary changes into the prototype block patterns, which resulted in a considerable cost savings. The display uses graphic z-buffering and PowerGraphics element faceting that provides a more realistic image. Image Courtesy of Cummins Engine Co. Inc.

Mapped Meshing

This display shows mapped meshing on a model of a diverter cassette of a fusion reactor. Numerous techniques exist to sweep, drag, and extrude an all quadrilateral mesh into a volume mapped mesh, or to "cut" the volume by the working plane into map-meshable (all hexagonal elements) volumes. Image courtesy of ITER Joint Central Team.



Examples

of ANSYS

Capabilities



Electromagnetics

This 3D model of a solenoid actuator is comprised of a laminated stator core and a cylindrical armature. The stator and armature are meshed independently and linked together with the ANSYS constraint equation interface feature. This feature allows the user to rotate the armature without remeshing in order to quickly study the 3D effects of the armature cut-outs on the device performance.

Contact/Superelement

Engineers at Komatsu Dresser used ANSYS nonlinear contact capabilities to analyze this model of a design for the front frame of a pay loader. A superelement, shown at the bottom of the model, was used to represent the pay loader axle which conveys loads to the frame. Gap elements and constraint equations were used to created the interface between the axle superelement and front frame to accurately model load transfer.

Thermal Analysis As part of a series of analyses conducted by Pratt and Whitney United Technologies, Inc., engineers performed a thermal analysis of the U.S. space shuttle's main engine tur-

bine outlet duct. The ANSYS automatic constraint generator was used to connect the regions with dissimilar mesh patterns.



Modal Analysis

This display shows a modal analysis of a truck radiator by a large automotive company. The Block Lanczos linear eigensolver, a member of the family of modal solvers, solves medium to very large problems quickly. Block Lanczos solved this image of an automotive radiator model in 1/25 of the time of the subspace solver, requiring about 1/3 of the disk space requirements.



Multiphysics

ANSYS demonstrated the coupled-field capability for Inductotherm by simulating an Inductotherm furnace. The furnace operates at a 3000 kW power level, 65 Hz frequency, and holds over 27 tons of molten steel. This image shows Emag flux lines and FLOTRAN velocity vectors indicating two toroidal flow eddies in this coupled-field, electromagnetic-flow demonstration.

Examples

of ANSYS

Capabilities



Viscoplasticity

Motorola Inc. used the viscoplastic analysis capabilities of ANSYS to optimize the manufacturing of hybrid power modules. This image represents the results of the ANSYS viscoplastic analysis on a quarter symmetry model of the copper baseplate. Motorola used the analysis to identify solders that fit its process requirements, utilizing Anand's model for rate-dependent nonlinear constitutive behavior.

MAL

Large Deflection

Zeppelin Luftschifftechnik GmbH used the ANSYS program to test the frame of the Zeppelin for strength and buckling stability under a variety of operating conditions. The rigid skeleton is designed to withstand the elements and allow year-round flight.



Examples

of ANSYS

Capabilities

Nonlinear Analysis

Researchers at Pittsburgh's Center for Orthopaedic Research at Shadyside Hospital and the Center for Medical Robotics and Computer-Assisted Surgery at Carnegie Mellon University use ANSYS to improve the quality of life. The precision to reproduce the complexity of biological structures, joints, and materials, and the strong nonlinear capabilities of the ANSYS program are necessary for the development of a computerized surgical simulator that can predict the outcome of patient-specific hip replacement procedures.





Design Optimization

Link Manufacturing in Sioux Center, IA teamed with ANSYS Support Distributor, DRD Corporation, to optimize the lateral control for a truck cab air suspension system. Using ANSYS integrated with Pro/ENGINEER, Link was able to reduce the weight of the mount almost 20 percent without significantly increasing maximum stresses. The ANSYS/ProENGINEER Interface allowed Link to maintain full associativity among key Pro/ENGINEER deliverables including the ANSYS finite element model, and part and assembly drawings. These images show the original and optimized versions of the control bracket. Image courtesy of DRD Corporation and Link Manufacturing.

Model Geometry Transfer

This solid model geometry was created with the Computervision CADDS program. Using ANSYS Connection for CADDS, this model was transferred directly into the ANSYS program, allowing design engineers to conduct ANSYS simulations on CADDS models without the use of external translators.





The ANSYS/LS-DYNA product can be used to effectively analyze dynamic impact, drop test, and material process simulation problems. In this analysis, a crashworthiness simulation of interconnected car seat rails was performed. As represented in the image, the large deformation dynamics, contact, and strain rate dependent plasticity capabilities of the ANSYS/LS-DYNA program were utilized. It is also possible to analyze fluid/structural interactions. The fluid elements can be used to solve for forces and thermal loads resulting from the flow. The user can apply the fluid flow information to a structural model to determine structural deformations and resulting stresses based on fluid forces acting on the structural model. Users can apply the structural deformation results to the fluid medium and recalculate the flow based on the new structural geometry. The user can conduct as many iterations as necessary to obtain a sequentially coupled solution. This coupled approach, which holds great promise for further coupling with other types of analyses, can be used for analyzing devices that operate on fluid/structural interaction, such as pulsation dampeners, aircraft wings, and artificial heart valves.



Figure 26

ANSYS/FLOTRAN CFD capabilities were used to display the magnitude of the velocity of air as it passes around a thrown football. Flow around the football is 60 MPH, 10/Revs/Sec, with a 15 degree angle of attack. Q-slices indicate air velocity vectors and pressure contours.

With the multiple species transport capability, users can monitor the transport of a mixture consisting of up to six different fluids, each of which may have distinct properties. This allows the mixing characteristics of different geometric designs to be evaluated for fluids with widely varying diffusion coefficients. The properties of the fluid carrying the species can be independent of the species or a strong function of their properties. The user can approximate the effects on a flowfield of entities that are not appropriate to model with geometric detail. These can be things that impede the flow (distributed resistance) or contribute to it (fan model). A screen in the middle of a flowfield is an example of distributed resistance. Resistances take the form of K-factors, friction factors, or permeability. The fan modeling capability allows the user to simulate the effects of a cooling fan or pump in a flowfield. The simulation can occur in a completely enclosed area, such as the flow inside a refrigerator; or in a completely open area, such as the flow that is passed over the cooling system of a refrigerator and exhausted.

Users can specify density, thermal conductivity, and viscosity in tabular form. Density can be specified as a function of pressure only.

ANSYS/FLOTRAN provides the ability to restart a FLOTRAN analysis from any results set, as well as giving the option of creating a CFD restart file, which minimizes restart time for large models.

Pipe Flow

Pipe flow analysis determines pressures, velocities, and heat-exchange characteristics of a fluid in a closed system, such as an automobile engine cooling system. This analysis type is applicable for any system with a constant flow rate of an incompressible fluid.

The finite element formulation for pipe flow analysis is as follows:

$$\begin{bmatrix} \mathbf{C}^{\mathrm{T}} & \mathbf{0} \\ \mathbf{0} & \mathbf{0} \end{bmatrix} \begin{pmatrix} \mathbf{T} \\ \mathbf{0} \end{pmatrix} + \begin{bmatrix} \mathbf{K}^{\mathrm{T}} & \mathbf{0} \\ \mathbf{0} & \mathbf{K}^{\mathrm{F}} \end{bmatrix} \begin{pmatrix} \mathbf{T} \\ \mathbf{P} \end{pmatrix} = \begin{pmatrix} \mathbf{Q} \\ \mathbf{W} \end{pmatrix} + \begin{pmatrix} \mathbf{Q}^{\mathrm{G}} \\ \mathbf{H} \end{pmatrix}$$

where:

- [C^T] specific heat matrix
- {T} nodal temperature vector
- {T'} time derivative of nodal temperature vector
- {P} nodal pressure vector
- [K^T] thermal conductivity matrix, including convection and mass transport effects
- [K^P] pressure conductivity matrix
- {Q} nodal heat flow vector

- {w} nodal fluid flow vector
- {Q^G} internal heat generation vector
- {H} vector of gravity and pumping effects (hydraulic head vector)

If fluid velocities and pressures are the only factors of interest, the temperature components of the formulation can be deleted. Pipe flow problems are nonlinear because the conductivity matrix changes with variations in the pressure differential. Therefore, the ANSYS program solves for the flow rate and temperature gradient through an iterative process in which the conductivity matrix is updated with each iteration to reflect the new pressure differential. The iterative process continues until the solution meets a predetermined convergence criterion for the number of iterations specified by the user.

The solution output is in the form of pressures at each node and flow rates through each element. Postprocessing functions can be used to produce graphics displays of pressures, flow rates, and temperature distribution.

Additionally, the thermal-fluid pipe element type can be used with the 3D surface effect element type to simulate a fluid mass flow about the exterior of a structure, including convection heat transfer effects. For example, users can simulate the thermal effects of air passing over a rotating turbine blade by modeling the air flow with thermal-fluid pipe elements and modeling the surface convection heat transfer to the air with 3D surface effect elements.

Acoustics

With ANSYS acoustic capabilities, users can study the propagation of sound pressure waves in a contained fluid medium or analyze the dynamics of a structure submerged in a fluid (Figure 27). For example, these capabilities may be used to determine the frequency response of an audio speaker, to study the sound distribution in a concert hall, or to predict the damping effects of water on a vibrating ship hull.



Figure 27

Using a harmonic response analysis, the ANSYS program calculates the sound pressure level of a rectangular acoustic driver as a function of driving frequency.

Acoustic analysis is made possible in the ANSYS program by special 2D and 3D fluid elements designed for this purpose. There are also infinite boundaries that represent the propogation of the waves into the surrounding medium. These elements are used to represent the fluid medium and the fluidstructure interface in the finite element model. Small density changes are assumed.

The finite element formulation representing the fluid-structure interaction in acoustics is:

$$\begin{bmatrix} \underline{M}_{S} & \downarrow & 0\\ \underline{Q}_{0}R^{T} & \downarrow & M_{F} \end{bmatrix} \begin{bmatrix} \underline{u}\\ p \end{bmatrix} + \begin{bmatrix} C_{S} & \downarrow & 0\\ 0 & \downarrow & C_{F} \end{bmatrix} \begin{bmatrix} \underline{u}\\ p \end{bmatrix} + \begin{bmatrix} K_{S} & \downarrow & -R\\ 0 & \downarrow & K_{F} \end{bmatrix} \begin{bmatrix} \underline{u}\\ p \end{bmatrix} = \begin{bmatrix} F_{S}\\ 0 \end{bmatrix}$$

where:

[Ms], [Cs], [Ks] respective structural mass, damping, and stiffness matrices [MF], [CF], [KF] respective fluid mass, damping, and stiffness matrices

pressure-displacement		
coupling matrix at the interface		
mean fluid density		
structural displacement,		
velocity, and acceleration vectors		
pressure and its time derivatives		
applied structural forces vector		

The solution is in the form of structural displacements and fluid pressures. Postprocessing options may be used to chart nodal displacements and pressures or to produce displays of pressure contours or structural deflections.

Coupled-Field Analysis

In the design of components under the influence of thermal, structural, fluid, electrical, or electromagnetic fields; there is often a need to consider the coupled influence of these fields. For example, a pressure vessel may require a structural analysis of the vessel for both internal pressure loads and thermal strain loads. Another application may be determining the eddy currents and skin effects of coupled electromagnetic fields in a transmission line or slot-embedded conductor in an electric apparatus. In both of these cases, coupled-field interaction can play a major role in the overall solution.

In the ANSYS program, field coupling can be achieved directly, through coupled-field elements, or indirectly, through sequential field analyses.

The direct method of coupling employs coupledfield elements in a single analysis. These elements have multiple degrees of freedom (spanning several fields) at each node to allow for continual cross-communication between the analytical disciplines involved. Sequential analyses are not required because the coupling is built into the governing equations through element matrices or element load vectors. An example of field interaction requiring the direct method of coupling is a circuit-fed solenoid actuator, where the circuit voltage and current are integrally coupled to the coil in an electromagnetic field simulation.

Unlike the direct method, the indirect method of coupled-field analysis involves two sequential analyses, each belonging to a different field. The two fields are coupled by applying the results from the first analysis as loads for the second analysis. The transfer of loads is accomplished through a single ANSYS command. For example, in a thermal-stress analysis, the nodal temperatures from the thermal analysis are applied as thermal loads in the subsequent structural analysis.

The indirect method is appropriate for one-way coupling situations, where the analyses are order dependent. In many cases, indirect coupling is not only more efficient than the direct method, but it also offers more flexibility since the two analyses can be performed independent of each other. Consider again the thermalstress example. The thermal analysis may be nonlinear and transient, while the stress analysis is static.

In the ANSYS program, coupled-field analysis is available for the following types of interaction: thermalstress, magnetic-thermal, magnetic-structural, fluid flow-thermal, fluid flow-structural, fluid flow-electromagnetic, thermal-electric, electromagnetic, circuitcoupled electromagnetic, and piezoelectric coupling (Figures 28, 29, and 30). Most of these interactions can be modeled by either the direct or indirect coupling method. The exceptions are piezoelectric, electromagnetic skin-effect analyses, and circuit-coupled electromagnetic field analysis for which the direct coupling method must be used. (A detailed discussion of piezoelectric analysis is presented in the following section.)







Figure 29

A step voltage input to the coil results in a time-history response of the current. The movement of the armature causes the characteristic dip in the response curve.



Figure 30

The armature displacement is tracked over time. The force on the armature from the magnetic field must first overcome a spring pre-load before the armature closes.

Piezoelectric Analysis

The piezoelectric capabilities in the ANSYS program are used to analyze the response of 2D and 3D structures to an AC, DC, or arbitrary time-varying electrical or mechanical loads. This analysis type may be applicable for components such as transducers, oscillators, resonators, microphones, and other electromechanical devices.

Four types of analyses are available for determining different aspects of piezoelectric response:

- Static Analysis for determining deflection, potential electric field, electric flux density, and stress distribution.
- Modal Analysis for determining natural frequencies and mode shapes.
- Harmonic Response Analysis for determining system response to harmonic loads (current, voltage, forces, etc.), including electrical admittance, impedance, electromechanical couplings, deflections, electric field, electric flux density, and stress distribution. This analysis can be performed at any phase angle relative to the input loading.
- **Transient Response Analysis** for determining system response to arbitrary time-varying loads (current, voltage, forces, etc.), including electrical admittance, impedance, electromechanical couplings, deflections, electric field, electric flux density, and stress distribution.

Users can model a piezoelectric structure using three ANSYS coupled-field solid elements. These elements allow for a variety of linear material property data input, such as complete 6 x 3 piezoelectric constants; isotropic, orthotropic, or anisotropic elastic stiffness or compliance constants; and diagonal real dielectric constants.

The ANSYS postprocessor can produce displays of actual deflections and mode shapes; and contour displays of electrical fields, electric flux densities, and stresses. Time-varying responses of a transient analysis can be viewed with the time-history postprocessor. Postprocessing capabilities allow additional calculations on all solution data for user-required parameters, such as admittance and impedance, electromagnetic coupling coefficients, and other solution values.

Substructuring

The ANSYS program contains extensive substructuring capabilities which may be used to improve solution run times or to increase modeling efficiency by reducing a group or set of elements to an equivalent, single, independent element. The basis of this technique involves matrix condensation, whereby the stiffness (or conductivity) and, if required, the mass (or specific heat) and damping matrices are reduced to a set of master degrees of freedom (MDOF). This process forms, in ANSYS terminology, a superelement.

Any element type or combination of element types may be used to generate a superelement. The only restriction is that the program assumes superelements are linear. (If nonlinear elements are included, they are treated as linear elements.)

To create a superelement, the user first defines a model of the region to be substructured along with MDOF that will characterize its behavior. The program then calculates the superelement matrices and writes them to a file. Once formulated, the superelement may be used in the ANSYS program in the same way as any other element.

A substructure can be used on any ANSYS analysis (Figure 31). It may contain only the defined superelement, or the superelement combined with other element types. The superelement may be placed directly in the model, or it may be repositioned by symmetry reflection, coordinate translation, or coordinate system transformation. Multiple superelements, as well as superelements within superelements, can be defined. The ability to graphically display the edges of a superelement enhances visualization of the overall model.

Superelements are most commonly used to separate or isolate certain portions of a model from the rest of the structure or to simplify repeated areas of a model. Some advantages of using superelements are listed below.

• A linear portion of a structure may be separated from the nonlinear portions. This effectively allows the nonlinear portions to undergo an iterative solution without subjecting the rest of the model to multiple solution passes.



Figure 31

The use of substructuring for repeated geometry can significantly reduce the total degrees of freedom needed in a model. This results in a savings of modeling effort and computer time. In this example, a 60 degree sector of the pulley is formed into a singlematrix superelement, then repeated to form the entire model.

- The modeling and solution time for a structure having repeated or symmetric linear element patterns (such as a gear) may be reduced. Rather than modeling the entire repeating pattern as a whole, one of the symmetric portions of the structure can be formed as a superelement which can then be repeated to form the complete pattern.
- With careful planning, several users can independently model sections of a structure, then bring the sections together to form a "full" model.
- A structure may contain components which, because of their design or manufacture, cannot be allowed to vary. By modeling such components as superelements, these "fixed design" portions of a model may be isolated from the rest of the structure, allowing them to remain unaffected by design or mesh refinement changes applied to the rest of the model. Substructuring is particularly useful in this respect when users employ ANSYS design optimization capabilities.

- Different types of ANSYS analyses may be performed on the same model without having to re-triangularize the stiffness matrix. Superelements are often used when two types of analyses (such as modal analysis followed by a linear transient analysis) are performed on the same structure.
- Stress calculations may be separated from displacement calculations in single-pass analyses. This is useful when the superelement stresses are not needed, but their displacements and effects on the rest of the model are desired.
- Flexible kinematic analyses can be solved more efficiently. Use of substructuring in a kinematic model is made possible by the superelement's large rotation capability.

Submodeling

Submodeling allows a particular portion of a model to be separated from the rest of the structure, re-meshed, and analyzed in greater detail. It can be a more efficient modeling method because the user can do a preliminary analysis with a coarser mesh and analyze finely meshed submodels only in areas of interest. Users can obtain more accurate information about a particular area of a structure without increasing the complexity of the entire model.

Submodeling can be used after an analysis has been performed on a full model and it is apparent that the results are not detailed enough in certain areas. This approach is useful when the user is not initially certain where high stress (or temperature, flux density, etc.) will occur in a structure or component. However, a more powerful application of submodeling is to plan ahead for its use in an analysis in order to reduce modeling and analysis effort.

The user creates a model with a finite element mesh that is fine enough to adequately represent gross interactions and to locate high-stress areas, but not necessarily fine enough to obtain accurate results in those areas. The advantage of a coarse mesh model is that it requires relatively little solution processing. Next, the user determines which areas of the model require further analysis through submodeling. For example, more detail may be needed in an area of high stress.



Figure 32

A relatively coarse finite element model of the pulley hub and spoke juncture is shown overlayed by a submodel. The displacements calculated from the coarse model are used as boundary conditions in the subsequent analysis of the submodel.

Having determined a region of interest, the user creates a new model (a submodel), which includes only that portion of the original structure. The finite element mesh in the submodel is made sufficiently finer than the coarse model mesh so that the results (stress, temperature, voltage, or flux density) will be accurately calculated within the submodeled region (Figure 32). The next step, which is the key to submodeling, is to transfer the behavior of the coarse model to the submodel boundaries, which represent cuts through the coarse model. Using the solution results from the coarse model, the ANSYS program determines the appropriate boundary constraints (displacements, temperatures, voltages, or potentials) and applies them to the submodel "cut" edges (Figure 33). Finally, the submodel is analyzed independent of the original structure, eliminating the need to reanalyze the entire model.

The submodeling technique offers the following advantages:

- Eliminates the need for complicated mesh transitions from fine to coarse regions in a model.
- Enables the user to study the effect of local geometric changes of alternate designs without reanalysis of the entire model.
- Allows reanalysis of areas of concern (such as high-stress regions) without prior knowledge of where these areas are located.
- Eliminates the need to initially include small geometric details (holes, fillets, etc.) that can be considered later in submodeling.
- Allows users to create solid element submodels from shell element coarse models.

Material Properties

The user can easily define any material property in the ANSYS program as isotropic and constant with respect to temperature. However, most material properties can also be defined as orthotropic and temperature-dependent.

Temperature-dependent properties are defined by one of two available methods. The first method involves defining a property-versus-temperature table. A set of temperature and property data points is input. Property values for the current element temperature are then obtained from this table by interpolation.

The second method of specifying temperature dependency is to define the material property as a fourth order polynomial function of temperature:

Property (T) = $A + B(T) + C(T)^2 + D(T)^3 + E(T)^4$,

where T is temperature and A, B, C, D, and E are input values representing coefficients of the polynomial. Not all coefficients need to be defined; for properties that are constant with respect to temperature, coefficients B through E are zero. If this form of data input is used, the property curve is converted by the ANSYS program to a temperature table similar to the one constructed directly in the first method.



Figure 33



The table containing the property-versus-temperature function is stored in the centralized ANSYS database, allowing the data to be manipulated in a variety of ways within the preprocessor. Database commands can be used to modify table entries. The data can also be written to a file to create a material property library, allowing the information to be used for other analyses or by other users. Finally, the user can display the property-versus-temperature curve.

Values for orthotropic materials are specified for the X, Y, and Z directions in the element or global coordinate system. If the property is defined in the X direction only, the Y and Z values default to the X direction values, thereby representing isotropic materials. For some structural and piezoelectric materials, a special constitutive matrix input can be used to represent anisotropic behavior. Material data are not restricted to isotropic or orthotropic properties. The ANSYS program can also represent anisotropic properties for selected elements.

Composite materials can be modeled by means of special multilayer shell and solid elements. These

elements allow stacking of isotropic or orthotropic material layers, with varying layer thicknesses and material orientations (Figure 34).

Nonlinear material properties are discussed in detail in the Structural Nonlinearities section, page 22.



Figure 34

Material properties can be nonlinear and functions of temperature, as this display of a nonlinear stress-strain curve at two temperatures, T1 and T2, for an analysis involving multilinear isotropic hardening plasticity illustrates.

The following shows the linear material properties that are available for each analysis type:

Material Properties

- Structural Analyses: Elastic (Young's) Modulus Coefficient of Thermal Expansion and Reference Temperature Poisson's Ratio Mass Density Coefficient of Friction Shear Modulus Material Damping
- Thermal Analyses: Specific Heat Enthalpy

Thermal Conductivity Convection (Film) Coefficient Emissivity

- Fluid Analyses: Viscosity Thermal Conductivity Density Specific Heat
- Electric Analyses: Resistivity Permittivity
- Electromagnetic Analyses: Material B-H Curve Permanent Magnet B-H Curve Relative Permeability Permanent Magnet Coercive Force
- **Piezoelectric Analyses:** Piezoelectric Matrix Elastic Stiffness Matrix Dielectric Matrix

Additionally, the user may retrieve material properties from a material library. The library may contain linear and nonlinear properties, and may also be temperature-dependent. The user may add and edit additional materials to suit analysis needs. The library may also be set up so that all users in a company can access the corporate library.

The ANSYS Element Library

The ANSYS element library consists of more than 100 element types. Many have options that allow further specialization of the element formulation in some manner, effectively increasing the size of the element library. Elements are categorized as 2D or 3D and may take the form of a point, line, area, or volume.

Both linear and quadratic (midside nodes) elements are available. Quadratic elements offer a higher accuracy for a given element mesh. However, the linear elements generally include extra shape functions to improve their accuracy. Midside nodes on any element edge can generally be deleted. Most 3D brick elements can be degenerated to prisms or tetrahedrons, and most 2D quadrilateral elements can be degenerated to triangles.

Most elements allow appropriate element loadings, such as pressures, temperatures, convections, etc. These are applied to the element which then calculates the corresponding load vector terms. Also, inertia loads (such as gravity) are available for most elements. Nodal loads (forces, temperatures, displacements, etc.) are allowed for all elements, as appropriate. An alternative method of applying loads to an element is with surface effect elements that can represent special loads such as surface tension radiation and foundation stiffness.

An element birth and death option, available for most element types, allows the user to activate or deactivate the contribution of an element to the matrices during the solution phase. Many structural and thermal elements also include an error estimation capability which allows the program to calculate the amount of solution error due specifically to mesh discretization (a key step in the adaptive meshing process, described in the Preprocessing section, page 9).

Several specialized elements allow the user to tailor a finite element model to specific needs. For structural analysis, the stiffness-damping-mass matrix element can represent a user-defined elastic kinematic response between two points in space. For more general purposes, the ANSYS program provides a user element capability that allows users to link their own element subroutines to the ANSYS object code. The user's element is then available, along with all other ANSYS elements, for any analysis. This user element capability provides significant flexibility and potential power to users with special requirements.

The ANSYS program allows users to choose between h- or p-element technologies. All of the ANSYS program's capabilities are available for h-elements. Let's explore the ANSYS program's capabilities with respect to p-element technology.

P-elements

The ANSYS program offers a comprehensive set of solid and shell p-elements that can be used for linear elastic structural analysis. This capability provides automatic solution accuracy control. P-solutions are costeffective because there is less user interaction with appropriate meshing.

P-elements allow the polynomial level to change from two to eight, depending on the solution accuracy desired. Because of the higher order solution representation used in p-elements, a coarse mesh may be used for the analysis. Additionally, because of data compatibility within the ANSYS program, existing h-element meshes can be converted into p-element models. This is especially true for models that use higher order h-elements. Users can control which elements may change their polynomial level, thereby reducing overall solution time. Additionally, p-element analysis does not require remeshing, thereby saving additional time.

Solution convergence is user-controllable and can consist of four independent criteria: global strain energy, local displacement, stress, and strain; or any combination of the four. Separate convergence tolerances can be applied.

Element displays are very realistic. Enhanced graphical images are possible because each element can be displayed using a multifaceted representation and displaying the actual curved geometry as well as the detailed stress contours. PowerGraphics visualization features are available for element and contour displays and are applicable to both p- and h-elements.

Postprocessing features that complement the p-element offering are very intuitive. The results for shell elements are displayed on the surface that is visible to the viewer with ANSYS PowerGraphics capabilities. This is also true for h-shells with PowerGraphics. There is no need to specify the top or bottom of a shell element for results output. Users can display the individual element polynomial levels and produce convergence plots for the various criteria chosen. Users can obtain solution results at a series of internal locations with the subgrid query feature. Element results are available in three forms: element centroid, nodal, and element subgrid (up to 25 locations on each quadrilateral element and 125 locations on a 3D solid element). The results picker probes data results for p-elements at the subgrid location closest to the user's mouse. H-elements have nodal query. Both p- and h-elements have element query capabilities. The data probe allows automatic viewing of the minimum and maximum solution values. Results are not averaged automatically at geometric discontinuities. The obvious case would be the edge between adjoining plates of different thicknesses. The user can also invoke non-averaging of results by material property.

The p-method supports the PowerSolver, allowing fast solutions for large problems, while minimizing hard disk space. It is an extremely attractive alternative to the h-method for linear elastic analysis.

Element Table

The element table at the end of this section provides a graphic representation and brief description of all available elements. ANSYS elements have other features that are not indicated on this summary table.

In the table, the elements are grouped into these categories:

- **Structural**: elements for static and dynamic stress analyses
- **Thermal:** elements for steady-state and transient heat transfer analyses
- Fluid: elements for analyses of fluid flow, CFD, acoustics, and contained fluids
- **Electromagnetics:** elements for static, harmonic, and transient magnetic analyses
- Electric Field: elements for electrostatic field analyses
- **Coupled-Field:** elements for analyses that involve one or more coupled-field effects (structural, thermal, magnetic, fluid, electric)
- General: elements that can be used in several analysis types
- **Infinite**: elements for field modeling and infinite media

The table briefly describes the element by:

- Title
- Name, consisting of a descriptive prefix and a unique number (e.g., BEAM3)
- · Typical number of nodes
- Applicable modeling space (2D or 3D)
- Degrees of freedom per node (DOF)

ANSYS elements possess other special capabilities including geometric, material, and element nonlinearities (typically, nonlinear elements display an abrupt change in stiffness when they experience a status change); element birth and death; and error estimation.

Abbreviations used in the table include:

UX, UY, UZ Translational Displacement DOF ROTX, ROTY,

- ROTZ Rotational Displacement DOF
- TEMP Temperature DOF
- PRES Pressure DOF
- AX, AY, AZ Vector Magnetic Potential DOF
 - VOLT Voltage DOF
 - MAG Scalar Magnetic Potential DOF
 - VX, VY Velocity DOF
 - ENKE Turbulent Kinetic Energy
 - ENDS Turbulent Energy Dissipation
 - CURR Current
 - EMF Potential Drop





















SURF22 8 nodes 3D space DOF: UX, UY, UZ, TEMP

Postprocessing

The postprocessing phase of the ANSYS program follows the preprocessing and solution phases. With this portion of the program, the user may easily obtain and operate on the results calculated in the solution phase through a very complete set of user-friendly postprocessing features. These results may include displacements, temperatures, stresses, strains, velocities, and heat flows. The output from the postprocessing phase of the program is in graphics display and/or tabular report form. Displays may be made on-line during an interactive postprocessing session at a graphics display device, or may be diverted for off-line printing. Because the postprocessing phase is fully integrated with the ANSYS preprocessing and solution phases, the user can examine results immediately.

During the solution phase, analysis results are written to the ANSYS database and to a file called, in ANSYS terminology, a results file. Results from individual substeps are stored as data sets.

The amount and type of data available for each data set are controlled by the type of analysis performed and the options that were set during the solution phase. For each load step of the analysis, the user may specify that a data set is to be written for every substep, the final substep, or some combination of the final substep and intermediate substeps. The user may also choose the extent that data groups, such as displacements, stresses, and reaction forces, are written (Figure 35).

Users can access data sets for postprocessing in two ways. They may examine the results of the entire model or any selected portion of the model for a particular data set using the general postprocessor. Or data for individually selected portions of the model, such as particular nodal displacements or element stresses, can be examined across multiple data sets with the time-history results postprocessor. When read from the results file, data is stored in the ANSYS database, allowing access to all input data (geometries, materials, load, etc.) during postprocessing. Database manipulations may be performed easily and, when used interactively, provide immediate graphics and/or listings of the results.



Figure 35

The mode shape of a thin membrane is determined with the ANSYS program, which displays results in several ways. This example shows displacement contours, the displaced shape of the perimeter elements, and a line plot of the displacements around the perimeter.

Postprocessing features that complement the pelement offering are very intuitive. Features such as contouring the p-level of the elements, obtaining the convergence history, and querying the p-element subgrid are all useful in investigating the analysis. The Q-Slice capability gives users results at any given plane within the model analyzed.

The ANSYS General Postprocessor

The general postprocessor may be used to examine results from any ANSYS analysis type. Data results may be selected, sorted, algebraically manipulated, combined with a data set from another substep, listed, or graphically displayed.

Several options exist for reading data sets from the results file into the database for postprocessing. The data set of interest can be identified by the load step and substep number, data set number, time, or frequency. If a time (in time-dependent analyses) is specified for which no results are available, the program performs linear interpolation on the two nearest data sets to calculate results at that time.

As in ANSYS preprocessing, a set of select commands allows portions of the database to be flagged for specific operations. Flagging may be done based on criteria such as displacements, stresses, geometric locations, pressures, and node and element numbers, among others. Selections may also be made by graphically picking the node or element with a mouse. Typically, the selection option is used to reduce results display time by limiting the active data set to nodes and/or elements of interest.

Tabular listings provide a convenient way of documenting analysis results for reports, presentations, etc. Sorting operations are available to organize tabular listings of stresses, displacements, pressures, voltages, or any other results item. Sorting options include arranging results in ascending or descending order, finding highest values, or sorting according to absolute value. To further customize listings for a report presentation, the user can control certain format features, such as the header at the top of the listing and the number of lines on a printed page.

Once the desired postprocessing data has been obtained (through selecting, sorting, algebraic manipulation, etc.), it can be displayed in many graphic forms. Contour displays show how a result (stress, for example) varies over the model. Typically, contours are in the form of lines, color bands, or isosurfaces (surfaces of constant value in three dimensions).

If discontinuities exist in the model, such as two materials joined together, an option exists to display the discontinuous stress at the boundary. For shell models, results for the top and bottom surface are displayed simultaneously with visible contours determined by the viewing direction. Field problems such as electromagnetics or fluids require inspection of field variations within a 3D model to validate the solution. Display capabilities such as particle clouds and gradient triad methods can be effectively utilized to visualize these fields. The user can query graphics results data and point to a location to obtain the numeric value at that point. Other types of graphic displays include the vector display, path plot, and particle flow trace. Vector displays use arrows to show the variation of both magnitude and direction of a vector quantity result, such as displacement or magnetic vector potential. Path plots are graphs that show the variation of a quantity along a user-specified path through the model. A particle flow trace, useful for fluid flow analysis, shows the path a particle travels in a flowing fluid.

Path operations can be used to map analysis data onto spatial paths described through a model. Once a result item is mapped onto a path, a tabular or graphical display can be used to visualize how that item varies along the path. In addition, mathematical operations (such as integration, differentiation, multiplication, dot, and cross products) can be performed among path items. This capability allows the user to calculate specialized quantities, such as J-integrals for fracture mechanics.

Another way to perform mathematical operations on results data is through element tables. Results can be read into an element table, which serves as a worksheet allowing arithmetic operations between its columns. Valid operations include (but are not limited to) addition, multiplication, division, exponentiation, and safety factor calculation.

Most results processing will involve the data set from one specific substep (such as the final substep of load step 1). The ANSYS program also provides load case combination capability, by which operations can be performed between two distinct data sets, termed load cases. Operations available for load case combination include addition, multiplication, square root, square-root-of-sum-of-squares, and maximum and minimum envelope comparisons. A typical example of load case combination is comparing and storing the maximum of two data sets.

One of the main concerns in a finite element analysis is the adequacy of the finite element mesh. The ANSYS program provides an error estimation technique that estimates the amount of solution error due specifically to mesh discretization. This technique, which the user can suppress, is available for linear structural and thermal analyses using 2D or 3D (solid or shell) elements. The error in energy norm calculated for each element can be viewed in the general postprocessor and used to determine portions of the finite element mesh that need to be refined. Using this error estimation technique and the powerful ANSYS Parametric Design Language (see ANSYS Parametric Design Language section, page 62), users can implement automatic adaptive mesh refinement to optimize the finite element mesh.

Mixed-mode stress intensity factors (KI, KII, and KIII) can be calculated for use in fracture analysis of models with linear material properties. When coupled with the PREP7 ability to automatically create a mesh around a crack tip, and/or with the path operations described previously, this feature produces a powerful fracture analysis capability.

The normally daunting task of calculating cumulative fatigue damage from the results data can be greatly expedited and automated by the fatigue capability available in the general postprocessor. Patterned after the American Society of Mechanical Engineers (ASME) Boiler & Pressure Vessel Code, it considers stress superposition and range counting for loadings of various origin. For stress-ranging that exceeds elastic behavior, a simplified elastic-plastic analysis is performed using penalty factors as suggested in the ASME Code. If applicable, stresses across a wall or section can be linearized to permit application of book-value stress concentration factors (Figure 36).

The Time-History Results Postprocessor

The time-history results postprocessor enables the engineer to select items such as nodal displacements, stresses, or reaction forces; and examine them over a time period or substep history of the analysis. These results can be reviewed as graph plots or tabular listings. Typically, this produces a curve such as displacement-versus-time. This feature is particularly useful for evaluating the results of transient structural or transient thermal analyses. Graphs illustrating one or more variables versus frequency (for harmonic analysis), or any other variable, may also be generated to assist in visualizing analysis results (Figure 37).





Figure 36

Path displays can graph results along a path through the model. The top figure compares the stresses through the fillet of the spoke and the hub for the full model and the submodel. The bottom figure compares stresses along the submodel boundary. Good correlation of boundary stress is one indication of the adequacy of the submodel.

Additionally, the time-history results postprocessor can perform algebraic manipulation of the curves. Variables may be added, subtracted, multiplied, and divided to create new curves. Other operations such as absolute value, square root, logarithm, exponential, and maximum and minimum determinations can be used. Differentiation and integration of the curves may be done to produce additional curves such as velocity and acceleration. It can also be used to generate response spectra (for use in a spectrum analysis) from time-history results.



Figure 37

The time-history postprocessor easily produces time-history graphs. Here, the temperature solution time-history of several nodes on a thermal model is monitored throughout a transient analysis.

ANSYS Parametric Design Language

The normal procedure for performing an analysis using the finite element method involves defining the model and its loading, obtaining a solution, and interpreting the results. If the solution results indicate that a design change is necessary, the geometry of the model must be changed and the process repeated. This procedure can be very costly and time-consuming, especially if the model is complex or many changes must be made.

ANSYS Parametric Design Language (APDL) gives the user the ability to automate this cycle by setting

up an "intelligent" analysis; that is, the program input can be set up to make decisions based on specified functions, variables, and selected analysis criteria. APDL allows for sophisticated data input, giving the user control over virtually any design or analysis entity such as dimensions, materials, loadings, constraint locations, and mesh refinement. APDL expands ANSYS capabilities beyond the realm of traditional FEA and into more advanced operations, including sensitivity studies, parametric modeling from parts libraries, innovative design changes, and design optimization.

The extent to which APDL can be employed to maximize the program's efficiency is limited only by the ingenuity of the user. For example, a company that manufactures gears may use the ANSYS program to analyze all of its new designs. The user can develop one generalized set of ANSYS input commands to describe the basic definition of a gear, including material properties, geometry, and other design parameters that gears may have in common. The user can quickly set up and perform an analysis for virtually any type of gear the manufacturer designs by changing specific values for the appropriate entities in this master set of input commands, and an analysis can be quickly set up.

Even more sophisticated use of APDL features in this example might be to automate the gear design process. The generalized ANSYS command file can be set up to prompt the engineer for detailed gear parameters (such as dimensions, material properties, number of gear teeth, pitch, loadings, etc.). Based on this data, the ANSYS program would create the gear model geometry and loadings, and execute the appropriate analysis. Furthermore, the program could be directed to retrieve the analysis results and decide if the gear design is acceptable based on defined limitations.

APDL consists of the following features, which can be used together or separately as desired:

- Parameters
- Array Parameters
- Expressions and Functions
- Branching and Looping
- Repeat Functions and Abbreviations
- Macros
- User Routines

All of these global control features allow the program to be customized to meet particular modeling and analysis needs. With careful planning and some ingenuity, the user can create a highly sophisticated controlling scheme that will maximize the program's efficiency for a particular realm of applications.

Parameters

APDL allows the user to define named variables (parameters) with values that the user specifies or the program calculates. Parameters may be defined at any point in an ANSYS session. In addition, they can be saved to a file for use in a future ANSYS session or for use in other programs and reports. The parameter capabilities provide a useful means of both supplying control to the program and simplifying data input (Figure 38).

A parameter may be defined as a constant value, the current value of a parametric expression, or a character string. Constant parameters are defined by simply assigning a value to an alphanumeric name. For example, the user may input the value of PI by issuing the command PI=3.14159. Once this parameter has been defined, the program substitutes the value 3.14159 when PI is used in an argument for any commands that follow. Constant parameters may also be determined by conditional tests. For example, the command, A=B<5.7 instructs the program to set the value of A to the current value of parameter B (if B is less than 5.7), otherwise, A will equal 5.7.

A parametric expression that is assigned to a parameter may include typical mathematical operations and/or FORTRAN functions. For example, the user can direct the program to obtain a model's length dimension by calculating the SRSS of two other model dimensions. If these two other dimensions have been defined as parameters X and Y, the user may input the command LENGTH=SQRT(X**2+Y**2). The program will then substitute this value wherever the parameter LENGTH appears. Valid families of operations include arithmetic, comparison, and nearest integer; and standard FORTRAN-type trigonometric, exponential, and hyperbolic functions.



Figure 38

APDL allows parameters to be used to define the geometry and/or other specifications of a model if regular changes or modifications of the design are needed. In this instance, the pulley is modeled parametrically using parameters to represent the rim thickness (THKRIM) and the spoke thickness (THKSPK) for a series of design studies. In addition to user-specified parameter values, ANSYS-calculated values may be assigned to parameters. A single command instructs the program to retrieve data from the model database, such as minimum or maximum node numbers and keypoint coordinates, or calculated stress and temperature values. Virtually any data items in the database may be assigned to parameters. This capability is especially important in the optimization process, as described in the Design Optimization section on page 66.

The task of retrieving ANSYS supplied data is made even simpler by the availability of alternate ANSYS functions, which return a value wherever the function is input without the need to assign it to a parameter. For example, the function NX(n) returns the X coordinate of node "n". These functions are commonly used as arguments in parametric expressions but can be used anywhere a numerical value is required.

Array Parameters

The type of data required for, and generated by, an engineering analysis is often more understandable when presented in tabular format. The availability of array parameters in ANSYS facilitates the processing of this type of data.

Array parameters are multiple-valued arrays that can be defined in matrix format. They may be 1D (one column), 2D (rows and columns), or 3D (rows, columns, and planes). The entries in an array parameter may have user-defined or ANSYS-calculated values. User-defined array parameters can be entered directly within an ANSYS session or read from an existing data file.

There are three types of array parameters. The first type consists of discrete numbers that are simply arranged in a tabular fashion. The second type, known as table array parameters, also consists of numbers arranged in a tabular format. However, this parameter type permits linear interpolation of values between the specified table entries. In addition, a table array parameter allows the index row and column to be filled with noninteger numbers. These features make the table array parameter a powerful tool for both inputting data and processing results. The third type of array parameter is a character array parameter, which consists of text strings.

Array parameters can be used to simplify data input. For example, a time-history forcing function can be input as a table array parameter with a minimum number of data points, and the ANSYS program can calculate force values required at times not specified in the defined array. Other applications for data input include (but are not limited to) response spectrum curves, stressstrain curves, and material-versus-temperature curves.

Another feature related to array parameters is the ability to do both vector and matrix operations. Vector operations (which apply to column vectors) include addition, subtraction, dot product, cross product, and more. Typical matrix operations, such as matrix multiplication, transpose calculation, and simultaneous equation solving, are also available.

At any point in an ANSYS session, array parameters (as well as other parameters) can be written to a user-specified file in FORTRAN real format. This feature can be used to write output files for use in other programs and reports.

Branching and Looping

An intelligent analysis requires a framework for decision-making. This framework is provided in the ANSYS program by the looping and branching features. Looping allows the user to avoid tedious repetition of commands, while branching gives the user global programming control and the ability to guide the program through an analysis.

Looping is achieved through typical DO-loop instructions that direct the program to repeat a series of commands. The number of passes through the loop is controlled by a counter or by other loop controls. These controls can direct the program to bypass portions of the loop or to exit the loop entirely based on the status of a given condition.

The branching feature uses traditional FORTRAN GO- and IF-type directives as a means of instructing the program to read commands in a nonsequential order. The GO command directs the program to an input line which is identified by a user-defined label. The IF command is a conditional that instructs the program to go to another line only if a given condition is satisfied. An IF-THEN-ELSE capability is also available which directs the program to perform one of several actions based on a current condition. IF commands may be used along with user-specified or ANSYS-calculated parameters to evaluate the condition.

Branching commands direct the program to make decisions based on virtually any model or analysis entity. This allows the user to perform parametric studies in which particular input quantities may be changed according to the value of a calculated quantity.



Figure 39

The design of a stamped sheet metal part might require decisions based on calculated quantities. An intelligent analysis of the stamping process is possible with APDL.

For example, in the postprocessing phase, the user can instruct the program to automatically produce stress contour displays if the stress value is below a certain level, and tabulate printouts if the value is above that level with branching commands. Another example of how branching can be used as part of an intelligent analysis is shown in Figure 39.

Repeat Functions and Abbreviations

Repeat functions simplify command input by eliminating unnecessary repetition of command strings. When the repeat command, ***REPEAT**, is entered in an input sequence, it re-executes the command immediately preceding it for a specified number of operations. The repeated command can be executed as it was input, or the arguments of the command can be incremented with each repetition. These functions can be used extensively to simplify model construction. Repeat functions may be used in model development to generate nodes, keypoints, line segments, boundary conditions, and other model entities.

With command abbreviations in the toolbar, users can simplify command input. Once defined, they can use an abbreviation anywhere in the command input stream.
Macros

A macro is a sequence of ANSYS commands that is saved to a file and may be executed at any other time in an ANSYS session. The user creates a macro file with a system editor or from within the ANSYS program. It may include any of the APDL features, such as parameters, repeat functions, branching, etc.

To create a macro from within the ANSYS program, the user instructs the program to copy a series of commands to a specified file. Macro files are automatically saved in the user's directory as they are created. At any point afterwards in the data input process, the user can direct the program to apply the macro file's command sequence.

Macros can be repeated any number of times within an analysis and can be nested up to ten levels. There is no limit to the number of macros that may be employed in any analysis. Macros that are used often can be grouped into a macro library file and applied individually in any ANSYS session.

One of the most obvious uses of a macro is to simplify repeated data input. For example, the same sequence of meshing commands may be required to create a mesh around several holes in the surface of a model. Typically, the string of commands needed to create the mesh would have to be repeated for each hole in the model. Instead, the user can create a macro containing all of the appropriate meshing commands. When meshing each hole, the user can instruct the program to run the macro file. Many other applications of this sort can use macros to eliminate repetitive command input. Users may place ANSYS picking commands (e.g., N, P) within macros.

When using macros, an alternative method of defining parameters is the *ASK command. This command asks for a parameter value by means of a userspecified message. The *ASK command is especially useful in automating the analysis of a structure for which basic characteristics (such as dimensions, material properties, etc.) may change from one design to the next.

An APDL feature that is commonly used within macros (but can be used in any file that is read into the ANSYS program) is the ***MSG** command. This com-

mand allows the user to write parameters and/or user-supplied messages to a user-controllable formatted output file. The message can be a simple note, a warning, an error, or even a fatal error (with the latter two capable of causing run termination). This allows the user to create custom reports from within the ANSYS program, or to generate formatted output files that an external program can read.

More powerful and sophisticated uses of macros are made possible by the program's ability to pass arguments into macros. This capability effectively allows the creation of input subroutines within the analysis.

Macros can be thought of as user-definable commands. If a command name is entered which the ANSYS program does not recognize, a search sequence is instituted in the directory structure. If a macro of the same name is found, it is executed. A user-specified path name can be implemented in the directory search, allowing commonly used macros to be conveniently grouped in a single directory for use in any ANSYS session.

The ANSYS program provides several prewritten macros, including the adaptive meshing macro (described in the Preprocessing section on page 9) and the animation macros. Other macros [such as the ANSYS American Institute of Steel Construction (AISC) macros for defining and evaluating AISC members] are normally announced in the ANSYS newsletter and made available to users upon request.

Animation macros in the ANSYS products include deformation with contours, Q-Slice with contours, Q-Slice with vectors, and isosurfaces. Users have the ability to save animation to a file, restore animation from a file, and control the rate of animation. Three dimensional devices support the Pixmap animation and display list animation. Animation macros enhance support for the ANSYS/LS-DYNA interface.

User Routines

Although not strictly considered part of APDL, user routines add to the flexibility of the program in a similar way by allowing the user to create highly specialized capabilities within the program. The open architecture of the ANSYS program enables the user to write a FORTRAN subroutine and link it to the ANSYS code. Possible user routines include:

- User-defined commands that enhance ANSYS capabilities
- A user-created element that can be employed in the same manner as other ANSYS elements once it has been defined
- Alternate failure criteria for the 100-layer composite shell and solid elements
- User-defined equations for creep and material swelling
- Alternate plastic material behavior specifications

Design Optimization

ANSYS design optimization is a computer technique that generates a series of finite element designs to obtain an optimal design. The engineer defines the criteria and boundaries of the design, and sets up the model parametrically, as in performing a parametric study. The optimization routine then controls and executes the analysis, deciding what new values to supply for the parameters to be used in each trial design. ANSYS design optimization permits virtually any aspect of a design to be optimized such as shape, stress, natural frequencies, temperature, magnetic potential, or discrete quantities, not just cost or weight as in more limited approaches. The ANSYS design optimization capability may be applied to any analysis and is the only design optimization available for electromagnetic and coupled-field analyses (Figure 40).

The design optimization process helps users measure and understand their design space. Optimization tools include the factorial tool, which scans all extreme points in design space; the gradient tool, which computes the gradient of the object function and state variables; and the sweep tool, which sweeps design space one design variable at a time. These tools provide the ability to do design sensitivity studies using derivative information, such as gradients of dependent variables with respect to design variables. It is described in terms of design variables, state variables, and an objective function. These terms are defined as follows:

- **Design Variables:** Design variables represent those input parameters of a design that are subject to change. They are usually geometric parameters such as length, radius, fillet radius, or material thickness; but may also be descriptors such as materials, locations of loads, or locations of constraints. The user must specify minimum and maximum limits, or side constraints for each design variable.
- State Variables: State variables are response parameters of the model that evaluate the design based on criteria specified by the user. Stresses, deflections, temperatures, or natural frequencies are typical state variables. Upper and/or lower limits are specified for each state variable, representing the engineering criteria that determine the feasibility of the design.
- **Objective Function:** The objective function, a single variable that characterizes the design, is the function that is to be minimized. Any quantity that can be expressed as an ANSYS parameter, including user-defined formulas (for example, one that relates production time to a machined fillet size), can be defined as the objective function. Other possible objective functions include total weight, a cost function, volume of material, or any appropriate performance criteria.

The user specifies the following: parametric input data for the initial design; design and state variables, including limits for each; and the objective function. The optimization routine selects new values of the design variables, analyzes the resulting design, evaluates the design against the state variables, and then uses the results of the evaluation to repeat the sequence in an effort to minimize the objective function.

Two methods of optimization are available in the ANSYS program: the subproblem approximation method and the first-order method.

With subproblem approximation, the program uses approximate functions that are obtained by curve fitting data points from previous trial designs. The approximate objective function is minimized with the sequential unconstrained minimization technique (SUMT) and used to produce the next design. The program represents the objective function as an unconstrained function by adding penalty terms to account for design and state variables constraints.

First-order optimization is an optimization technology that uses derivative information, such as gradients of dependent variables, with respect to design variables. The program computes the gradient and forms an unconstrained objective function through adaptive penalty functions. Search directions are formed during each iteration, and a line search strategy is adopted to minimize the unconstrained problem.

Of the two techniques, subproblem approximation is more efficient in finding an optimized design; however, first order is the more robust technique. The ANSYS program allows the use of both techniques sequentially. A typical example is the use of subproblem approximation to narrow the design space followed by the application of the first order technique to hone in on the best design.

In addition to the traditional optimization procedure that finds an optimum design, ANSYS offers a series of optimization tools.

Optimization tools help users measure and understand the design space of their problem. Since minimization may or may not be a goal, an objective function is not required for use of the tools. However, design variables must be defined. The following tools are available.

- Single Loop Run: This tool performs one loop and produces one FEA solution at a time. The user can do "what if" studies with a series of single loop runs, setting different design variable values before each loop.
- Random Design Generation: Multiple loops are performed with random design variable values at each loop. The user can specify maximum number of loops and desired number of feasible loops. This tool is useful for studying the overall design space and for establishing feasible design sets for subsequent optimization analysis.
- Sweep Generation: Starting from a reference design set, this tool generates several sequences of design sets. Specifically, it varies one design variable



Figure 40

Design optimization can be applied to any ANSYS analysis. In this case, the pulley is optimized for the highest first frequency, while the rim and spokes are subject to a thickness constraint. For this elementary 2D example, no limits were placed on stress or on any other response quantities (i.e., no state variables).



Figure 41

In ANSYS design optimization, design variable parameters are revised repeatedly in order to minimize the objective function while staying within the limits of the state variables for defined engineering criteria. Adaptive meshing can be an integral part of the optimization process.

> at a time over its full range, using uniform design variable increments. This tool makes global variational evaluations of the objective function and of the state variables possible.

- Factorial Evaluation: This statistical tool generates design sets at all extreme combinations of design variable values. This technique is related to the technology known as design of experiment that uses a two-level, full, and fractional factorial analysis. The primary aim is to compute main and interaction effects for the objective function and the state variables.
- **Gradient Evaluation:** At a user-specified reference design set, this tool calculates the gradients of the objective function and states variables with respect to design variables. With this tool, users can investigate local design sensitivities. They can graphically view results of these design explorations, which provide valuable insight into the design and design sensitivities.

Since the model can also be described parametrically in geometric terms of lines, curves, areas, and volumes; the solid modeling and automatic meshing features of the ANSYS preprocessor add even more functionality to design optimization. A parametrically defined solid model gives the user precise control of the geometry when doing shape optimization. In addition, parameters can control mesh density, allowing wide variations in the geometry from design cycle to design cycle without compromising mesh quality.

Another way to maintain mesh quality during optimization is to let the ANSYS program find the optimum mesh by means of the adaptive meshing procedure. By this method, the program automatically generates and optimizes the mesh for each trial design, resizing the mesh repeatedly until an acceptable mesh discretization error is achieved. To increase efficiency, the user can selectively call the adaptive meshing routine when it is most beneficial to optimization (for example, when the discretization error of a trial design mesh has exceeded a certain level). Figure 41 shows how the adaptive meshing loop works within the design optimization cycle.

Third-Party Program

ANSYS, Inc. ensures that users receive the most comprehensive offering of engineering tools available today by partnering with best-of-class software vendors. Integrated products provide users with several advantages over point solution software that only addresses one type of problem. Benefits include customized, cost-effective solutions with a single, intuitive interface. Because the ANSYS family of products has compatible data structures, results from one program, such as ANSYS/Structural, can be used in another, such as ANSYS/Multiphysics. Enterprisewide engineering is easy to implement when companies have the right tools, such as ANSYS programs, that are a strategic part of an efficient product design and manufacturing cycle.

The ANSYS Enhanced Solution Partners (ESP) program supports high-quality, vertical application developers that utilize ANSYS as a platform to create custom products. This innovative program gives developers the opportunity to transfer their industry knowledge into working commercial software. Developers create their own customized solutions from within the ANSYS environment using a suite of development tools that include macros, user interface design language (UIDL), user programmable features, and ANSYS parametric design language (APDL).

The application programmer's interface (API) is a key feature of the ESP program that provides a flexible environment to transform industry-specific knowledge into usable design software. Developers use the API to create their own niche applications. ESP members can create custom pre and postprocessors, as well as data libraries.

The ESP program supports a variety of software developers with applications currently underway. For additional details, see the ESP Third Party Software Directory on the ANSYS HomePage or contact your ASD or ANSYS, Inc. for more information.

The ANSYS CAD Relations Program

Easy-to-use, transparent access to CAD data is essential



Figure 42

Complex solid model geometry developed in Unigraphics and transferred directly into ANSYS using ANSYS connection for UnigraphicsTM.

in getting products to market faster by increasing productivity and the accuracy of new product designs. The ANSYS program shares data with many leading CAD vendors, and ANSYS, Inc. is actively working with many more to develop improved means for our mutual users to work productively. The nature of the solutions generated from these relationships between ANSYS and CAD companies is dictated by the needs of our mutual users. ANSYS, Inc. is providing users across all industries with easy access to state-of-the-art analysis capabilities and more ways to use FEA to shorten their production cycles.

CAD integration services provide solutions for today's existing CAD products. New products and parts are analyzed without requiring the costly and time-consuming efforts involved in rebuilding the model in the analysis system. Users gain direct access to the high-level functionality in the ANSYS program such as nonlinear behavior, electromagnetics, and CFD.



Figure 43

The solid model geometry data for this end plate was originally created in a CAD system, then transferred into ANSYS using the IGES translator provided in an ANSYS auxiliary processor. Model courtesy of Hewlett-Packard Mechanical Design Division. Services exist to transfer geometry from a CAD system to ANSYS. ANSYS Connection tools are available for CADDS[®], Pro/Engineer[®], and Unigraphics[™], and are in development for SolidWorks[®] (Figure 42).

ANSYS maintains compatibility with its entire product line and across all products based on its technology. This ensures that you do not run into a "brick wall" by using design simulation products that are not capable of advanced analysis. No matter what your application or CAD system, ANSYS is your design simulation solution.

NURBS-based model geometry data can be transferred among many programs through open standards, such as IGES (Figure 43). The current IGES specification allows precise transfer of complex geometry including NURBS trimmed surfaces, surfaces of revolution, and tabulated cylinders. Once within the ANSYS program, the model data can be analyzed, changed, and written out to an IGES file, suitable for transfer back to the CAD or engineering program that generated the original model.

ANSYS analysis of a model created on a CAD system must follow the IGES data transfer procedure codes. Closed surface splitting for B-spline and surface revolution is available. Boolean operations within IGES have improved. ANSYS has the capability to make those Boolean adjustments automatically, saving the user time.

Future ANSYS development initiatives are using emerging standards, such as PDES/STEP (the Product Data Exchange Specification/Standard for the Exchange of Product data) and the .SAT file format from ACIS Technologies.

In addition to NURBS-based data transfer, ANSYS, Inc. also provides translators to the ANSYS program for engineering programs. Data files from these programs may contain only finite element data such as node location, element connectivity, and even material properties or boundary conditions. Once the translation has taken place and the data are expressed in terms of ANSYS preprocessing commands, the full capabilities of the preprocessor are available for further refinement of the model. Bi-directional translators are available for CSA/NASTRAN, UAI/NASTRAN, and MSC/NASTRAN, as well as Algor, COSMOS, PATRAN, I-DEAS, and ABAQUS.

Quality Assurance of the ANSYS Program

Ongoing Development

The ANSYS program is continually revised and updated to enhance existing features, add new FEA capabilities, and make use of advances in computer hardware. Program upgrades are released regularly to current licensees with maintenance. This ongoing series of enhancements ensures that engineers have leading-edge ANSYS technology for their analyses.

Quality Assurance

Software quality and reliability are issues of primary importance for end-users and developers. The professional who is ultimately responsible for a design requires quality software for engineering computations. An effective quality assurance (QA) program for software has many elements, including a commitment from management; a dedicated, highly-qualified staff; and strict adherence to technical procedures. The creation of quality software is an ongoing process at ANSYS, Inc. that continues throughout the software development cycle.

ANSYS, Inc. maintains a philosophy that addresses the quality of the ANSYS program as an obligation. This approach is rooted in the company's commitment many years ago to meeting the American Society of Mechanical Engineer's (ASME) Nuclear Quality Assurance (NQA) standards.

Software quality assurance has been a fundamental process for the ANSYS program since the 1970s. ANSYS, Inc. created the first QA error report and error correction system used within the finite element software community. This system of customer notification still serves as a standard by which others are measured.

In 1983, the QA Department was formed within the company with the sole responsibility of quality assurance. Since that time, software verification testing has expanded to include a set of over 5,500 verification tests. Acceptance testing on all of the additional computer systems supported consists of a subset of over 1,500 tests. These testing procedures are highly automated to minimize human error in the reviews.

The Corporate Quality Department instituted regression checks on error corrections; created tests for modules, libraries and elements; and initiated graphics and FE meshing tests. Internal quality audits are performed yearly to maintain and enhance the effectiveness of the QA program.

In 1995, ANSYS, Inc. became the first design analysis software developer to achieve ISO 9001 certification. This certification, an international quality system standard, proves ANSYS, Inc.'s outstanding performance and affirms a continued commitment to quality.

QA Services Available

ANSYS, Inc. offers three quality assurance services that represent different levels of support to assist customers in meeting their internal QA requirements. Investing in the Testing Agreement, the QA Agreement, or auditing rights, enables your company to run more rigorous, onsite testing of the ANSYS program when you undergo systems changes, such as upgrading the operating system, changing processors, or installing different math or vector libraries.

Customer Services

Customer Services at ANSYS, Inc. provides service and support through multiple programs that ensure the customer's success. The programs combine technology, usability aides, and customer support to allow ANSYS to meet a wide range of customer requirements. ANSYS, Inc. and ANSYS Support Distributors (ASDs) provide these services, which range from hotline support to training courses.

TECS

The Technical Enhancements and Customer Support (TECS) program is one of the many service offerings

that provide specific results-oriented problem solving to customers. The TECS program provides a multitude of value-added services including continuous technological improvements and quality, centralized technical support. A worldwide problem-tracking system provides ANSYS technical support and ASDs with a centralized database for all customer support issues and product enhancements requests. Customers receive high-quality, personalized maintenance of their license through seamlessly integrated support.

The Company's customer support system consists of the global network of ASDs that provide localized assistance, and senior-level corporate staff located at the Company's headquarters. The combined expertise and experience of ASDs and corporate staff ensure a maximum return on investment for customers.

The customer support telephone hotline, available at both ASD and ANSYS, Inc. locations, is highly valued by ANSYS users. Licensees receive immediate assistance from experienced professionals.

Consulting

ASDs, certified with adherence to ISO 9001 guidelines, are authorized to license and support the ANSYS program based on strict technical requirements. Like ANSYS, ASDs commit to providing high-quality services around the world. Skilled consulting engineers are available through the ASD network for specialized consulting jobs. For hard-to-solve problems, ASDs provide the best technical support in the industry.

Program Customization Services

ANSYS, Inc. created the Program Customization Services group to tailor the ANSYS program to meet individual engineering requirements. Highly qualified development staff address customized programming and analysis needs. Services include CAD integration, results processors, solver customization, systems performance and tuning, specialty elements, specialty optimizers, new material models, and general custom programming.

The Program Customization Services group; using core ANSYS as a platform for modeling, analysis, and result evaluation; offers specialized programming for customer-specific applications that can be directly integrated within the ANSYS program. Through these services, engineers in multiple industries gain access to the powerful and robust analysis tools available in ANSYS products.

Development is currently underway for interfacing major CAD packages with ANSYS. These interface customization services represent just one example of how the Program Customization Services group can mold and shape the ANSYS program to meet specific customer requirements. These interfaces make ANSYS directly accessible from within CAD software environments; improving design speed and quality, eliminating rework and data transfer delays, and expanding access to ANSYS technology.

Training

Both ASDs and ANSYS, Inc. offer a complete series of ANSYS training programs. A three-day introductory seminar provides attendees with a comprehensive overview of ANSYS capabilities and familiarizes them with operational techniques. Advanced seminars offer a more in-depth study of subjects including dynamics, heat transfer, solid modeling, nonlinearities, and substructures. Seminars on specialized topics such as design optimization, undersea structures, magnetics, and user elements are also regularly presented.

Documentation

The ANSYS documentation set, distributed on-line, provides a complete program description, data input information, and explanations regarding two ways to perform ANSYS operations (using the GUI menus or commands). This set includes Analysis Guides, one for each analysis discipline with example scenarios; Command and Element References; Theory Manual; Workbook; and Verification Manual.

On-line documentation enables a powerful hypertext-based HELP system that includes a descriptive outline of new product features, and helps users correctly complete an analysis. The user can retrieve detailed information on program functions, commands, and procedures, often through one or two mouse clicks. Users can get text, diagrams, and other program information by selecting a hypertext block in the main HELP index, or by using the system's word search capability. Users type in the topic for which they need information (e.g., nonlinearities), and the program does the rest.

In addition to the on-line documentation set, ANSYS provides an ANSYS Operations Guide which is an introductory manual describing how to use the ANSYS GUI and how to perform basic ANSYS operations.

Users can also order User Guides which are in-depth treatments of specific features, such as fracture mechanics, design optimization, or composite structure elements. These notes are used in conjunction with ANSYS training seminars, but are also available by order. Theory, methodology, command explanations, and examples are included.

PQ

The Productivity Quotient (PQ) Assessment was created to give design organizations a tool for assessing the efficiency of their particular design process, and to help software developers identify areas for improvement while getting the most out of their technology investment.

Developed by D. H. Brown Associates, a leading engineering technology analyst firm, PQ measures an organization's effectiveness against industry standards regarding applications of computer-aided design and simulation technologies. Product development managers and design specialists answer questions in 25 key categories that assess the productivity of their processing.

Evaluating and assessing productivity is the only way to determine the effective application of simulationbased design software. Productivity is the ultimate test of a software's success.

ANSYS News®

ANSYS, Inc. publishes a quarterly technical periodical. *ANSYS News*, a magazine available to all users and others interested in the ANSYS program, provides tips on program use, reviews new ANSYS capabilities, includes important ANSYS, Inc. business news (including the CEO message), and previews upcoming seminars and user group meetings.

Index

A

Abbreviations, 65 AC electromagnetic field analysis, 34 Acoustic analysis, 40 Adaptive meshing, 12 Anand model. 25 Animation, 4 Anisotropic behavior, 25 Anisotropic material properties, 45 **ANSYS Element Library**, 46 ANSYS News, 73 ANSYS Parametric Design Language (ADPL), 62 ANSYS program, general description, 1 ANSYS Support Distributors (ASDs), 71 ANSYS/AutoFEA 3D, 7 ANSYS/ED, 7 ANSYS/EMAG, 7 ANSYS/FLOTRAN, 7 ANSYS/LinearPlus, 7 ANSYS/LS-DYNA, 7 ANSYS/Mechanical, 7 ANSYS/Multiphysics, 7 ANSYS/PrepPost, 7 ANSYS/ProFEA, 7 ANSYS/Structural, 7 ANSYS/Thermal, 7 Arc-length method, 21 Areas, 10 Array parameters, 64 Automatic time stepping, 23

В

Bauschinger effect, 25 Besseling model, 25 Bilinear isotropic hardening, 25 Birth and death (element), 47 Block Lanczos, 18 Boolean operations, 11 Bottom–up solid modeling, 10 Boundary conditions, 14 Branching, 64 Buckling analysis, 20

С

CAD packages, 5, 69 Classical bilinear kinematic hardening, 25 Combination element, 28 Components, 9 Composite materials, 45 Computational fluid dynamics (CFD), 37 compressible flow, 38 incompressible flow, 37 laminar flow, 37 thermal/fluid. 38 turbulent flow. 37 Conduction, 30 Conjugate Gradient solver, 38 Conjugate Residual solver, 38 Constraint equations, 14 Constraints. 14 Consulting, 72 Contact surface elements, 28 Contact surfaces. 28 Control element. 28 Convection, 30 Convergence checking, 31 Cooling effects, 37 Coordinate systems, 9 Coupled-Field analysis, 41 Coupled-Field elements, 41 Crank-Nicholson time integration method, 35 Creep, 26 Customer support, 71

D

Damped eigenvalue analysis, 18 Database, 5 DDAM spectrum analysis, 20 Degrees of freedom (DOF), 14 Density, 46 Derivative results, 37 Design optimization, 66 Design variables, 67 Dialog boxes, 3 Direct generation, 13 Direct integration. *See* Newmark time integration method Direction of flow, 37 DO–loop, 64 Documentation, 72 Drucker-Prager plasticity behavior, 24 Drucker-Prager yield criterion, 25 Dynamic kinematic analysis, 29

E

Eigenvalue, 18 Eigenvalue buckling, 21 Eigenvalue extraction, 18 Electric circuit analysis, 36 Electric current conduction, 35 Electric field analysis, 13, 35 Electric field elements, 48 Electromagnetic analysis, 41 Electromagnetic elements, 48 Electromagnetic field analysis, 33 Electromagnetic skin-effect, 41 Electromechanical interaction, 42 Electrostatics. 36 Element nonlinearities, 28 Elements. 46–58 Enhanced Solution Partners (ESP) program, 69 Enthalpy (in phase change), 32 Equation solver, 14 Error estimation, 47 Expressions and functions (parametric), 62

F

File format, 5 Finite element analysis (FEA), general description, 1 First-order optimization, 67 Flexible-body kinematics, 29 Fluid analysis, 37 Fluid elements, 48 Fluid flow analysis, 37 Fluid flow – structural analysis, 41 Fluid flow – thermal analysis, 41 Fluid – structure interface, 40 Frontal solver, 14 Functions (parametric), 65

G

General elements, 48 General postprocessor, 59 Geometric nonlinearities, 26 Graphical user interface (GUI), 2 Graphics, 4 Graphics window, 3 Guyan reduction, 17

Η

Hard copy graphics, 5 Harmonic magnetic field analysis. *See* AC electromagnetic field analysis Harmonic response analysis, 19 Heat flux, 37 Heat transfer, 37 Heat transfer analysis, 30–33 Heating effects, 37 HELP. *See* Hypertext–based HELP system HPGL, 5 Hyperelastic elements, 53 Hyperelasticity, 26 Hypertext–based HELP system, 4, 72

Ι

IGES, 2, 69 Incomplete Cholesky Conjugate Gradient (ICCG) Solver, 15, 33 Incompressible flow, 37 laminar flow, 37 turbulent flow, 37 Infinite boundary elements, 33 Infinite elements, 48 Input window, 2 Integration time step, 17 Interface elements, 28 Interfacing programs, 69 ISO 9000, 71 Isotropic hardening, 25 Isotropic materials, 45

J

Jacobi Conjugate Gradient solver (JCG), 15

Κ

Keypoints, 10 Kinematic hardening, 25 Kinematics, 29

L

Large deflection, 27 Large deflection analysis, 27 Large strain, 17, 26–27 Layered elements, 52 Lift and drag forces, 37 Linear (eigenvalue) buckling, 21 Linear actuator element, 30 Linear transient dynamic analysis, 17 Lines, 11 Load case combination, 60 Load data, 14 Looping, 64

Μ

Mach number, 37 Macros, 65 Magnetic – structural analysis, 41 Magnetic – thermal analysis, 41 Main menu, 2 Mapped meshing, 12 Master degrees of freedom (MDOF), 14 Material nonlinearities, 24 Material properties, 45 Matrix element, 58 Maxwell's equation, 33 Menus, 2 Meshing, 11 Modal analysis, 18 Mode shapes, 18 Mode superposition, 17 Model verification, 11 Mooney-Rivlin model, 26 Motif standard, 2 Mouse picking, 11 Multilinear elasticity, 26 Multilinear kinematic hardening, 25 Multilinear isotropic hardening, 25

N

Natural frequencies, 18 Newmark time integration method, 17, 23, 30 Newton-Raphson method, 17, 24, 31 Nodes, 14 Non-Uniform Rational B-Spline (NURBS), 2, 11, 70 Nonlinear buckling, 21 Nonlinear damper element, 28 Nonlinear elements, 28 Nonlinear spring element, 28 Nonlinear static analysis, 22–23 Nonlinear transient dynamic analysis, 22–23 Nonlinearities, 22–29

0

Objective function, 67 Optimization, 66 Output window, 3

Ρ

P-element, 47
PQ, 73
Parameters, 63
Parametric language. See ANSYS Parametric Design Language
Parametric modeling, 68
Parametric studies, 64

Phase change analysis, 32 Piezoelectric analysis, 42 Pipe flow analysis, 39 Plasticity, 24 Postprocessing, 59 Postscript, 5 PowerGraphics, 4 Power spectral density (PSD), 20 Prandtl-Reuss flow equation, 25 Preconditioned Conjugate Gradient solver (PCG), 15 Preconditioned Conjugate Residual solver (PCCR), 38 Preprocessing, 9 Pressure coefficient, 37 Pressure drop, 37 Prestressed analysis, 27 Processors, 5 Program Customization Services, 72

Q

Q-slice, 4 Quality assurance, 71

R

Radiation, 30, 31 Random vibration analysis, 20 Reinforced solid element, 28 Repeat functions, 65 Response spectra, 20 Response spectrum analysis, 20 Results, 59 Results file, 59 Revolute joint element, 29 Rigid-body kinematics, 29 Rubber-banding, 4

S

Scalar potential method, 34 Seismic analysis, 20 Shape optimization, 68 Shell element with wrinkle option, 28 Skinning, 11 Snap-through buckling, 22, 27 Software Z-buffering, 4 Solid modeling, 10 Solution method, 13 Solution phase, 13 Specific heat, 32 Spectrum analysis, 20 Spin softening, 26 Stability (buckling) analysis, 20 State variables, 67 Static analysis, 15 Static electromagnetic field analysis, 33 Static kinematic analysis, 30 Steady-state thermal analysis, 30 Stiffness. 14 Stream function, 37 Stress stiffening, 26 Structural buckling analysis, 20 Structural dynamic analysis, 16 Structural elements. 48 Structural nonlinearities, 22 Structural static analysis, 15 Submodeling, 44 Subproblem approximation method, 67 Subroutines, 66 Substructuring, 43 Superelement, 43 Support services, 71-73

Т

Technical Enhancements and Customer Support (TECS), 71 Tension-only/compression-only spar, 28 Thermal analysis, 30 Thermal conductivity, 46 Thermal elements, 48 Thermal–electric analysis, 41 Thermal–stress analysis, 41 Thermal–structural analysis, 32 Thermal/fluid analyses, 37 Third–party program, 69 TIFF, 5 Time – history analysis, 17 Time – history postprocessor, 18 Time – varying electromagnetic fields, 34 Toolbar, 3 Top-down solid modeling, 10 Training, 72 Transient dynamic analysis, 17 Transient dynamic analysis, 17 Transient electromagnetic field analysis, 34 Transient thermal analysis, 31 Translators, 70 Tri-Diagonal Matrix Algorithm (TDMA), 38

U

User-defined element, 47 User routines, 66 Utility menu, 2

V

Vector potential, 33 Velocities, 14 Velocity distribution, 37 Vibration analysis, 20 Viscoelastic element, 53 Viscoelasticity, 26 Viscoplasticity, 26 Volumes, 10 von Mises Yield criterion, 24, 25

W

Work hardening, 25 Working planes, 11

Y

Yield criteria, 24



ANSYS, Inc.

201 Johnson Road Houston, PA 15342-1300

ansysinfo@ansys.com T 412.746.3304 F 412.746.9494 *Toll Free USA and Canada:* 1.800.WE.R.FEA.1 *Toll Free Mexico:* 95.800.9373321

Regional Offices:

North America Izera@ansys.com T 810.585.5020 F 810.585.5730 Pacific Rim jtung@ansys.com T 412.873.3086 F 412.746.9699 Europe bbutcher@ansys.com T 44.1.734.880.229 F 44.1.734.880.360

http://www.ansys.com



ISO 9001 CERTIFICATION INCLUDES ALL COMMERCIAL PRODUCTS

ANSYS, FLOTRAN, ANSYS/ProFEA, and The Productivity Quotient are registered trademarks and ANSYS/Multiphysics, ANSYS/Mechanical, ANSYS/Structural, ANSYS/LS-DYNA, ANSYS/LinearPlus, ANSYS/Thermal, ANSYS/FLOTRAN, ANSYS/Emag, ANSYS/AutoFEA, ANSYS/PrepPost, ANSYS/ED, DesignSpace, Powered by ANSYS, and ANSYS Designer Series are trademarks of SAS IP, Inc. All other products, brand names, or company names are the property of their respective holders.