ANSYS®

Fundamental FEA Concepts and Applications

A Guidebook for the Use and Applicability of Workbench Simulation Tools from ANSYS, Inc.

Forward – Demystifying FEA

There are many high-quality FEA Fundamentals books available, “Building Better Products With Finite Element Analysis” by Vince Adams and Abraham Askenazi is one such highly recommended book (available from Amazon.com). The main purpose of this primer is to provide the reader with enough basic understanding of FEA fundamentals to understand how ANSYS Workbench Simulation works with a bias toward how to apply this tool in order to reap the most reward from its implementation.

A definition of terms, analysis types, assumptions and limitations will be covered and, where meaningful, additional explanation as to “Why is this important?” and “How and where would it be used?” will be provided. In addition, a section will cover when to apply ANSYS Workbench Simulation and when to look to more advanced tools such as ANSYS Professional or other ANSYS configurations that are offer more detailed analysis capabilities but also require more training and expertise.

In Perspective

FEA, for this discussion, is simply a tool to better understand how a design will perform under certain conditions. All analysis are approximations accounting for the most significant variables among many. What is almost as useful as obtaining an “exact” answer with regard to “reality,” which might take a highly trained specialist a great deal of time performing what is often called a “detailed” analysis, is the *trending* and *behavioral* information that can be used to help predict and improve one’s design (a “basic” or “fundamental” analysis). This approach substantially improves the overall engineering process. A very simple example is: how and where does this design bend? Noting how and where parts move can be often be just as helpful as knowing if it moves 1 inch or 1.0625, especially early in the design. Accuracy is important and ANSYS is known for quality, accurate products. Within its scope of functionality, ANSYS Workbench Simulation is perfectly capable of performing detailed analysis on par with traditional FEA methods in a fraction of the time. For any analysis, detailed or fundamental, it is vital to keep in mind the nature of approximations – study the results and test the final design. Proper use of ANSYS Workbench Simulation will greatly reduce the number of physical tests required while allowing the designer/engineer/analyst to experiment on a wider variety of design options and improve the end product.

Product Terminology

ANSYS Workbench in and of itself is not a product, rather it is a product development platform and user GUI built for analysis needs with the objective of providing elegant next generation functionality and intelligent automation to the engineering community. Workbench can be thought of as a “kernel” from which applications are built in a way similar to how solid modeling kernels such as ACIS or Parasolid are used.

ANSYS Workbench Simulation is the primary application module used in the Workbench environment and it is available in several product configurations. DesignSpace V6.0 was the first product built and delivered using Workbench technology in September of 2001. Today, the Workbench Simulation module can invoke any product license available from ANSYS DesignSpace through ANSYS Multiphysics; the higher the license level the more analysis capability exposed. The term “Workbench Simulation” is used throughout this document and will typically reference the product DesignSpace but may also reference functionality only available when using an ANSYS Professional or higher-level license.

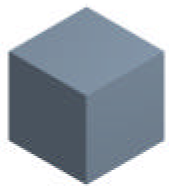
While other ANSYS Workbench product modules are available (DesignXplorer, DesignModeler, FE Modeler), this document will focus only on the Simulation module, regardless of license level used.

What is FEA?

Finite **E**lement **A**nalysis is a mathematical representation of a physical system comprising a part/assembly (model), material properties, and applicable boundary conditions {collectively referred to as *pre-processing*}, the solution of that mathematical representation {*solving*}, and the study of results of that solution {*post-processing*}. Simple shapes and simple problems can be, and often are, done by hand. Most real world parts and assemblies are far too complex to do accurately, let alone quickly, without use of a computer and appropriate analysis software.

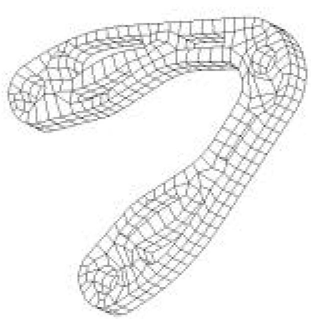
Pre-Processing

To do this, FEA software typically uses a CAD representation of the physical model and breaks it down into small pieces called finite "elements" (think of a 3-D puzzle). This process is called "meshing." The higher the quality of the mesh (collection of elements), the better the mathematical representation of the physical model. The primary purpose of an element is to connect nodes with predictable mathematical equations based on stiffness between nodes; the type of element used often depends upon the problem to be solved. The behavior of each element, by itself, is very well understood. By combining the behaviors of each element using simultaneous equations, one can predict the behavior of shapes that would otherwise not be understood using basic "closed form" calculations found in typical engineering handbooks.



• A block (element) with well defined thermal, mechanical, and modal behaviors.

• Example of a simple part whose structural behavior would be difficult to predict using equations by hand.



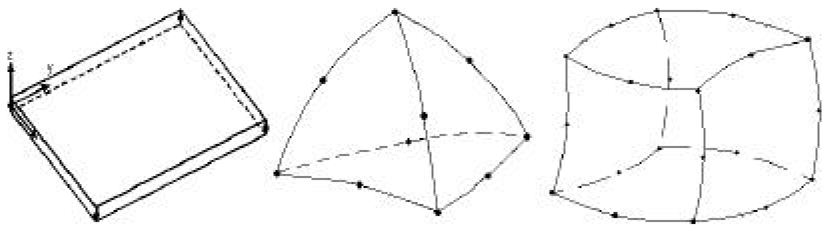
• The same part broken into small blocks (meshed into elements) each with well-defined behaviors capable of being summed (solved) and easily interpreted (post-processed).

There are many different types and classes of elements, most created for specialized purposes (cable, piping, beams, truss structures, e-mag, etc.). A one-dimensional element represents line shapes, such as beams or springs. A 2D element, also known as a quadrilateral element, will represent triangles and squares. 3D elements represent solid shapes and are usually in 2 basic shapes: brick (hexahedrons or "hex") and pyramids (tetrahedrons or "tets").

Examples of applications for specialized elements would be: scaffolding consisting of connecting 1D line elements. Car bodies and other stamped or formed sheet metal parts are typically very thin relative to their overall size and are usually best represented by 2D shell/plate elements. Many thin shapes can, and are, meshed with 3D solid elements but at the cost of increased processing time and sometimes a loss in accuracy because of the special formulation of 2D shell elements. The tradeoff is that, in order to mesh with 2D shell elements, there is often significant modification and preparation required to the CAD geometry in order to obtain a meshable surface model, or models in the case of an assembly. In other words, the pre-processing requirement increases substantially. The guidelines are: can it be meshed and solved in 3D solids (sometimes the resultant model is simply too large)? If yes and the user is not looking for ultimate accuracy but only trending and behavioral information, then 3D solid meshing is often appropriate due to the human time savings. 3D elements are ideal for thick and chunky parts and assemblies such as engine blocks, machine components, etc.

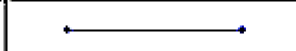
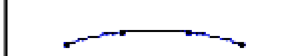
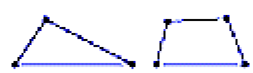
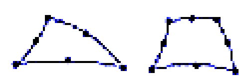
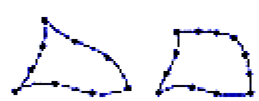
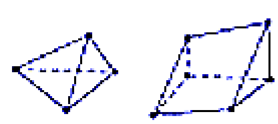
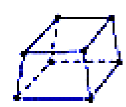
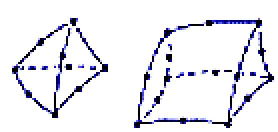
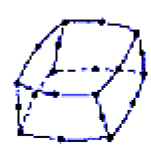
General purpose, modern mechanical FEA programs typically use a select set of elements chosen for their versatility, robustness, and their overall contribution to product ease of use. Workbench Simulation uses several primary element types and will default to high-order (10 node quadratic) tetrahedral (H) elements (SOLID 187 in ANSYS-Speak) for solid model geometries if they are not sweepable, in which case high-order (20 node) brick elements (SOLID 186) are employed. On closed surface models, "quad-dominate" (4 node) shell elements (SHELL 181), are used providing both accuracy and efficiency while being suitable for the robust automatic meshing algorithms used in Workbench Simulation. And, for part-to-part interaction within assemblies, high-end surface-to-surface contact elements (CONTACT 170/174) are used. For mixed beam/shell models and for spot-weld features, beam elements are employed (BEAM 188). ANSYS Workbench Simulation applies these various element types automatically.

Common Element Types Used in ANSYS Workbench Simulation



• 4-Noded Shell   10-Noded Tetrahedral   20-Noded Hexahedral

Other Common Element Types

Dimension	Degree	Element Shape	Element Type
1D (Line)	Linear		Beam, Truss
	Quadratic		Beam
	Cubic		Beam
2D (Area)	Linear		
	Quadratic		Plane stress Plane strain Plate, Shell
	Cubic		
3D (Volume)	Linear		
	Quadratic		

Each element is comprised of 2 or more "<u>nodes</u>" which help define its shape as well as to convey physical reactions from one element to the next. The "finite" in FEA comes from the fact that there are a known number of elements in a finite element model. The solver adds up the individual behaviors of each element to predict the behavior of the entire physical system.

Other aspects of the pre-processing phase involve identifying material properties and environmental conditions the design will be subject to. These conditions include various forms of physical forces (loads, pressures, moments, etc.), thermal loads and conditions (temperature, conductivity, convection, etc.), and constraints (fixed, pinned, frictionless/symmetrical, etc.). Some Workbench Simulation pre-processing fundamentals:

ANSYS Workbench Simulation Meshing

ANSYS Workbench Simulation provides 2 forms of automated meshing: Fully automatic and Manually Directed Automatic. Both forms employ a fault-tolerant philosophy meaning that, if a problem occurs, at least 12 attempts of automatic trouble-shooting are made before the mesher fails and tags the area of difficulty with a label. Manually directed means that the user may specify meshing overrides on specific areas of a part (edge(s), face(s)) or the baseline mesh density on entire parts that differ from other parts within the assembly, either for accuracy or efficiency purposes. These changes remain associative.

Workbench Simulation Material Data

Structural and thermal material data are defined, modified, and used in Workbench Simulation for structural and thermal analyses. Material properties include Young's modulus, Poisson's ratio, density, coefficient of thermal expansion, and thermal conductivity. The latter quantity, conductivity, can be temperature-dependent.

Workbench Simulation Convection Data

Temperature-dependent film coefficients are defined, modified, and used in Workbench Simulation for thermal analysis. Many categories of film coefficients can be devised to take into account laminar, turbulent, forced, and natural convection conditions, as well as various geometric configurations

Workbench Simulation Fatigue Material Data

Materials in the Workbench Simulation material library may include a fatigue stress-life curve populated with data from engineering handbooks. Fatigue data has been pre-populated for the Structural Steel and Aluminum Alloy material data files from the MIL-SPEC handbook (MIL-HDBK-5H) (http://analyst.gsfc.nasa.gov/FEMCI/links.html). Since material data is crucial to accurate fatigue results, Workbench Simulation readily allows the input of this information to new or existing materials by hand or through load history files. The Fatigue Tool will then use the information in the stress-life curves for each material in the model when calculating life, damage, safety factors, etc.

Workbench Simulation Pre-Processing Extensions

Many additional pre-processing capabilities can be realized by using the ANSYS Preprocessing Command Builder in which the user has greatly expanded capabilities to modify the analysis. APDL commands can also be added to the command stream to extend capabilities even further. The ANSYS Preprocessing Command Builder is enabled in Workbench Simulation when using an ANSYS Professional or higher-level license.

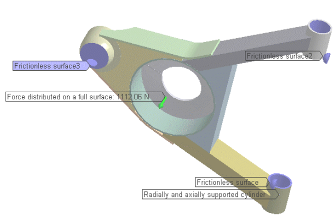
More sophisticated analysis types such as fluid flow, Micro Electro-Mechanical Systems (MEMS), and transient (time dependent) have many additional environmental and material property considerations, which require additional expertise to properly employ and are beyond the scope of discussion here.

ANSYS Workbench Solving

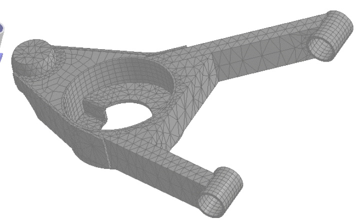
ANSYS Workbench employs 3 of the ANSYS solvers and automatically chooses the most appropriate or efficient solver for the job at hand. In addition to linear/static, ANSYS Workbench performs Coupled analysis types (thermal-stress, stress-modal, thermal-stress-modal) as well as some limited non-linear analysis types (thermal with temperature-dependent material properties and convection, geometric/contact with contact supporting lift-off). All solver settings and iteration propagations from one solve step to the next are performed automatically.

Post-Processing (Interpretation Of Results)

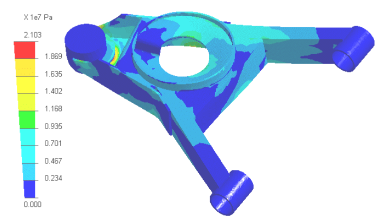
The output of a solver is generally a very substantial quantity of raw data. This quantity of raw data would normally be difficult and tedious to interpret without the data sorting and graphical representation referred to as post-processing. Post-processing is used to create graphical displays that show the distribution of stresses, strains, deformations, temperatures, and other aspects of the model. Interpretation of these post-processed results is the key to identifying areas of potential concern (weak areas in a model), areas of material waste (areas of the model bearing little or no load), or valuable information on other model performance characteristics (thermal, modal) that otherwise would not be known until a physical model were built and tested {*prototype*}.



• CAD Assembly with Loads and Supports



• Meshed Assembly (Hex and Tet Elements)



• Stress Plot of Solved Problem

The post-processing phase of FEA is where the most critical thinking must take place, where the user looks at the results (the numbers vs. color contours, movements, etc.), and compares results with what might be expected. It cannot be stressed enough that it is up to the user to determine if the results make sense, to be able to explain the results based upon engineering "common sense." If the results are other than expected, one must search until an explanation can be found before the results can be fully trusted.

ANSYS Workbench Post-Processing

A select set of results is available for the user to view and interrogate. The set of results are fundamental stress, strain, deformation, thermal, shape optimization, and fatigue. Interrogation involves several options:

- ***Scoping***: Isolating parts from an assembly, or faces from a part and displaying results only on those areas.
- ***Slicing:*** Drawing a slice plane through any results figure then rotating the model so that the slice can be seen. Multiple slices through any one figure are allowed, as is the ability to drag a slice through the part.

A select set of results is available for the user to view and interrogate. The set of results include fundamental stress, strain, deformation, thermal, shape optimization, mode shape, and fatigue. Interrogation involves several options:

- ***Scoping***: Isolating parts from an assembly, or faces from a part and displaying results only on those areas.
- ***Slicing:*** Drawing a slice plane through any results figure then rotating the model so that the slice can be seen. Multiple slices through any one figure are allowed, as is the ability to drag a slice through the part.
- ***Probe:*** Dynamically display the results at cursor tip. Left mouse button allows for "probe tags" of results to be displayed on the model.
- ***Interactive Legend:*** Allows user to dynamically change color bands for results display.
- ***Deformed Shape:*** Several settings from undeformed to 5:1 automatic factor deformation. The automatic factor is calculated to ensure a visible deformation in the displayed model, whether the factor is 0.01 or 100.

Many additional post-processing plots may be obtained using the ANSYS Postprocessing Command Builder in which POST1 results are made available. APDL commands can also be added to the command stream to embed any other desired graph, output, or POST26 results. The ANSYS Postprocessing Command Builder is enabled in Workbench Simulation when using an ANSYS Professional or higher-level license.

General Types of Engineering Analysis

Structural analysis will be either linear or non-linear. Linear model analysis assumes that the material does not plastically deform (permanent deformation). Non-linear models consist of separating contact conditions (contact with lift-off), when stressing material past its elastic capabilities into the plastic range, or bending greater than 10% of model length (large deformation). At this point, material properties change and stresses in the material will vary with the amount of deformation. Vibrational analysis is used to test a model for its natural, resonant frequencies (to avoid a rattling muffler during idle conditions, or other failure such as happened with the Tacoma Narrows bridge) random vibrations, shock, and impact. Each of these incidences may act on the natural vibrational frequency of the model, which, in turn, may cause resonance and subsequent failure.

Fatigue analysis helps designers to predict the life of a model by showing the effects of cyclic loading on the part/assembly. Fatigue is responsible for approximately 80% of all structural failures.

Heat Transfer analysis shows the conductivity or thermal properties of the model. This may be either steady state or transient transfer. Steady-state transfer refers to constant thermal properties that yield linear heat diffusion and where time is not a factor in either the loading of thermal properties or in reading results at various time increments. Non-linear will usually involve time, radiation, and/or changing thermal properties in the material or through convection.

Shape optimization considers the physical load paths through a model and categorizes elements into 3 load-bearing levels: Significant, Marginal, and Low. If an element carries significant load, it will not be considered for identification for material removal. Shape optimization is not based on stress results, though it is analogous to some degree: areas of high stress are likely significant load bearing elements and will not be identified for possible removal. Results are purely visual and conceptual giving the user intuitive clues on where material may be reduced in the model through use of his CAD system. Workbench Simulation does not perform the actual material removal. Modified models should still be run through Stress analysis for verification.

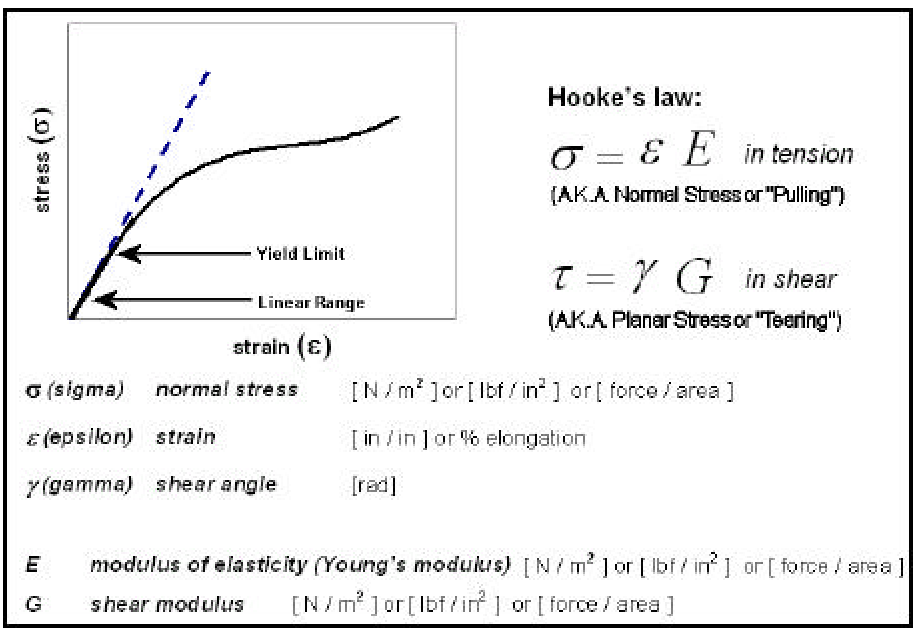
Vibration analysis consists of several types: fundamental or natural frequencies, with or without pre-stressing forced vibration, noise-vibration-harshness or randomly induced vibration such as would be experienced in an earthquake. Workbench Simulation helps determine if a part or structure is susceptible to environmental vibrations, as with might be experienced with rotating equipment or other induced vibrations (picture an opera singer breaking a glass, or the Tacoma Narrows bridge collapse).

Harmonic analysis is used to determine the steady-state response of a linear structure to loads that vary sinusoidally (harmonically) with time. The results encompass the structure's response at several frequencies and provide graphs or animations of a response quantity, such as displacement versus frequency. Harmonic analysis is supported in Workbench Simulation when using an ANSYS Professional or higher-level license.

Stress and Strain – The Basics

Stress occurs when any physical object is subject to loading and is expressed in the form of Force per unit area such as PSI for Pounds per Square Inch. There are two fundamental forms of stress: Normal (pulling) and Shear (tearing). Strain is a dimensionless quantity calculated as the ratio of deformation to the original size of the body – the amount of stretching that the object undergoes due to loading.

The relationship between stress and strain, and the equations that govern them are shown in the following graph:



• Basic Engineering Concepts - Stress vs. Strain and Hook's Law.

Notice that the first part of the curve is very linear. The slope of this curve is what is defined as the Young's Modulus of the material, and is well understood for common material types. In this linear range, stress and strain are proportional and therefore straightforward to calculate and where the term "linear stress" comes from.

Three dimensional stresses and strains build up in many directions. A common way to express these multi-directional stresses is to summarize them into an "Equivalent" stress, also known as the "Von Mises" stress.

All objects will have a stress limit depending upon the material used and is referred to as Material Yield. If steel had a yield limit of 40000 psi, any stresses above this limit would result in some form of permanent deformation. (Notice in the graph above where the stress-strain curves start to diverge, this is the yield limit of the material.)

If a design is not supposed to permanently deform by going beyond yield (most cases), then the maximum allowable stress in this case would be 40000 psi. A Factor of Safety can be calculated as the ratio of the maximum allowable stress to the equivalent stress (Von Mises) and would need to be over 1 in order for the design to be acceptable (below 1 means there will be some permanent deformation). In Workbench Simulation, stress templates are provided that automatically calculate both the stress and sheer Factor of Safety for ductile and brittle materials. Ductile materials generally refer to steel, aluminum, brass, and other like metals that tend to stretch more than fracture. Brittle materials refer to glass, concrete, cast iron, ceramics, and certain types of hardened steels. The factor of safety used for brittle materials in Workbench Simulation is the Mohr-Coulomb stress theory.

Factor of safety results in Workbench Simulation are very useful since they will immediately point out areas of potential yield whereas equivalent stress results will always show “red” in the highest area of stress regardless of how high or low the value. Since a factor of safety of 1 means the material is essentially at yield, most designers will strive for a safety factor of between 2-4 based on the highest expected load scenario. It should also be noted that, unless the maximum expected load will be frequently repeated, some areas of the design that go into yield does not always mean the part will fail. One must, as always, use engineering common sense to evaluate the situation.

Users of FEA

For the most part, users of FEA may be grouped as ***advanced***, ***intermediate***, or ***fundamental***.

Advanced

Traditionally, an analyst (a specialist in FEA) would spend considerable time learning how to correctly apply and use FEA software for advanced problems involving non-linear material properties (composites, highly elastic, etc.) non-linear geometry conditions (what happens after material yield and beyond to failure), time conditions (drop tests, crash tests, thermal radiation), and dynamics (earthquake analysis), to name just a few. Typically, an analyst spends most of their time using FEA software and very little in a CAD modeling system. Analysts are almost always degreed engineers who do little actual design but are highly regarded within their company for their expertise. This user will perform all analysis types from fundamental through advanced. Because of their expertise, most of their efforts are spent on advanced analysis, leaving little time for fundamental analysis problems usually encountered by engineers.

> If a company has a dedicated in-house analyst, then they will normally be consulted during a purchase of up-front simulation tools for fundamental analysis needs. In these cases, Workbench Simulation products (DesignSpace in particular) are approved more often than any other tool in their class, especially after thorough benchmarking involving real-world assemblies and problems because of the automation, robustness, accuracy, and the ANSYS solvers employed.

Intermediate

The intermediate user of FEA is also almost always a degreed engineer who may split his or her time between product design (with or without CAD) and analysis software solving problems both intermediate and fundamental. In the scale of users, there are more of these intermediate users than there are analysts. ANSYS Workbench Simulation addresses the entire fundamental range and some of the mid-range of these users needs in the areas of fatigue, manual mesh refinement controls, shell meshing, harmonic analysis, and non-linear contact conditions.

Fundamental

The bulk of FEA is used for fundamental analysis problems addressed by ANSYS Workbench Simulation; linear/static stress, deflection, factor of safety, thermal, and modal. The typical user of Workbench Simulation (usually DesignSpace) is an occasional user of FEA, spending most of his/her time designing in CAD, and should have good "engineering common-sense." This user may or may not be a degreed engineer but must have a good understanding of how the product being designed will be used so that it may be properly simulated. While just about anybody can use this class of tools, software cannot replace good critical thinking: Is the product behaving as expected (did I use the right loads/supports)? Are the answers within expected ranges (did I use the proper materials and unit system)? In short, does it all make sense and how can the results be used to properly evaluate the design.

In What Areas Can FEA Help?

Simulation, especially early in the design cycle, can bring dramatic benefits to almost any engineering discipline, including ***Product Design***, ***Machine Design***, ***Automotive Component Design***, and ***Aerospace*** when creating new designs and/or modifying existing designs. Examples of situations where early design simulation may help:

Over-Designed Products

Almost all manufactured products from all design disciplines include excessive material of some sort because adequate simulation was not done early enough in the concept and design stages to identify wasted material. If simulation was done at the end of a product design cycle, it may have been too late and costly (in time) to change the design and start over.

- *Costs incurred*: For high-volume products, count percentage of material over-use (typically 10-15%), any extra shipping/handling charges based on weight, and manufacturing processes affected (more efficient tooling may be possible).

- *Analysis types*: Factor of Safety (is the part 5 or more times over material yield?), Shape (conceptually identify areas of possible material removal), Stress (to verify design, identify areas of weakness, identify areas of over-design), and Fatigue (product life verification).

- *Example case study*: Home Gardner; saved 20x cost of Workbench Simulation on first project alone.

Under-Designed Products

Few of these make it past the first prototype stage and if they do, it bodes ill for the designer and the company that produced the product; hence, most products are over-designed. Of course, if a design does not pass the first prototype stage, that means that at least one more prototype must be made, and sometimes several.

- *Costs incurred*: Count the cost of each prototype; fabrication costs, personnel time, lost opportunity, increased time to market, lost market share, etc. Each prototype cycle represents enormous costs to the company that few like to talk about *yet it also represents the area of highest potential savings*.

- *Analysis types*: Factor of Safety (is the part approaching material yield?), Stress (to verify design, identify areas of weakness), Deformation (does it bend too much?), Thermal (is enough heat being dissipated, is an area at an unacceptable temperature?) and Fatigue (product life verification).

- *Example case study*: Buck Knives; received a pre-holiday buying season order for 100,000 units because ANSYS Workbench helped identify areas of weakness and allowed them to design a final product in a single prototype from the typical 3 prototypes usually performed.

Miscellaneous Product Performance Issues

Is the product subject to vibration regularly incurred in its operating environment? Does it bend too much or not enough or require too much force to bend it a specific distance (snap-fits)? Are there thermal conditions to consider in areas of the design that may be causing undue stresses or temperatures outside acceptable ranges? Do thermal and mechanical stresses play a part in determining what the natural harmonizing frequencies might be (thermal-mechanical pre-stressed modal)?

- *Costs incurred*: Excessive prototyping, customer dissatisfaction due to product feel or noise problems due to products natural frequencies matching environment vibrations bringing harmonic resonance into play (imagine an engine compartment running at idle RPM's creating harmonic conditions for your product which then starts to rattle).

- *Analysis types*: Modal (to identify products natural frequencies and make design/material modifications if they are too close to environment operating conditions), and Deformation (how much it will bend and where, how much force (reaction force) is required to make it bend a particular distance?)

Engineering Assumptions for ANSYS Workbench

ANSYS Workbench makes the following assumptions about calculated results:

Linear Structure
Calculated displacements are directly proportional to the load applied to a part or assembly. Linear behavior results when the slope of the stress-strain curve in the elastic region (measured as the Modulus of Elasticity) is constant.

Elastic Structure
A part or assembly returns to its original shape when the loads are removed. Elastic behavior results when the stress in a part corresponds to the elastic region of the material's stress-strain curve.

Small Displacements
The calculated displacements are small in comparison to the principal dimensions of the part. For example, if studying the deflection of a beam, the calculated displacement must be significantly less than the minimum cross-section of the beam. The small displacements assumption is related to small strain analysis, in which normal strains are required to be very small compared to one.

Linear Contact
The contact conditions between two or more parts in an assembly are treated in a linear fashion. For stress, shape, and modal analyses, only bonded and frictionless (no separation) contact conditions are supported. As such, ANSYS Workbench does not treat nonlinear friction and

nonlinear open-close (gaping) contact. For thermal analysis, heat flows across contact surfaces under the assumption of no thermal contact resistance.

Non-Linear Contact
The optional Optima module enables non-linear lift-off contact conditions ("Frictionless" and "Rough") to exist between two or more parts in an assembly. These conditions require substantially more CPU resources to compute and are best used in smaller models whose behaviors are well understood.

Undamped, Small Amplitude Vibration
A part or assembly vibrates in such a way that its deformations are very small compared to its size and damping effects are ignored.

Steady State Heat Transfer
Transient effects are ignored and all results and loads are time-independent.

General Assumptions and Limitations

In terms of capabilities, it is important not to keep capabilities of any software and the experience of its user in proper perspective and to use the right tool at the right time. One cannot use a hatchet to cut down a forest (ANSYS DesignSpace on a MEMS problem) – it can't be done. Nor should one drive a Sherman Tank down a sidewalk (ANSYS Multiphysics on a simple linear-static stress problem) – it is wasteful and a poor use of machine, software, and human resources. Following is a simplified reference table of which product license level is appropriate and when.

Problem Type	Class	Problem Example
Structural: • **Linear Stress, Deflection, Safety Factor**	**DS**	*Almost any strength of materials problem from single part to large assembly. Will any part of the design reach material yield?*
• **Buckling**	**DS**	*Compressive failure of structures, mostly of a "long" nature.*
• **Snap-fits, press-fits, bolted connections** • **Large deflection or what happens beyond yield, permanent deformation.**	**AP/A**	*Diving board, permanent deformation of any material. When will it truly break? Intentional crimping of materials.*
Thermal: • **Steady State**	**DS**	*Printed circuit board, engine components, "at rest" temperature, time is not a factor.*
• **Radiation**	**AP**	*No physical conduction/convection – spacecraft, components in a vacuum.*
• **Transient (time dependent)**	**A**	*Knowing the rate of temperature change or applying different energies/temps at certain times. Knowing how quickly a cooling fan will bring temperatures into range.*
Modal: • **Natural Frequencies (free & prestressed)**	**DS**	*Is a part sensitive to its environment; engine compartment, or other components exposed to vibrations?*
• **Dynamic (applied frequencies, change over time)**	**AP**	*Earthquakes, other induced frequencies where time may also be an important factor (noise/vibration/harshness testing).*
Materials: • **Homogenous Materials (Isotropic – Metals, most Plastics, Ceramics)**	**A**	*Most applications*
• **Non-homogenous, highly elastic (Anisotropic – Fiberglass, reinforced concrete, rubber)**	**A**	*Rubber, gaskets and seals where compression is an issue, composite materials.*
Time: • **Not a significant factor**	**DS**	*Loads steadily applied, results given represent final "at rest" state, not the transition to the final state.*
• **Time-dependent (load happens at X time), Need to know how much time (thermal, how long to reach a temp/state)**	**A**	*The rate at which heat is applied to a system compared with specific area and overall temperatures at various times within a heating cycle before "steady state". Applying vibrational forces over time (earthquake*

Class: DS = ANSYS DesignSpace AP = ANSYS Professional A = ANSYS Structural or higher

Problem Type	Class	Problem Example
Solid Assemblies	**ALL**	*Assemblies of more than 1 part, all created using 3D solids.*
Surface-model support (1 part)	**ALL**	*Non-3D solid surface (sheet) model of a single part.*
Surface-model assemblies	**ALL**	*Mix and match 3D Solid models with surface models in an assembly.*
Dynamics: • **Modal analysis, Pre-stressed Modal analysis**	**DS**	*Determine a structure's vibration characteristics: Automobile tailpipe assembly could shake apart if its natural frequency matched that of the engine. A turbine blade under stress (centrifugal forces) shows different dynamic behaviors. A printed circuit board will be subject to both thermal and mechanical stresses yet must not have a natural frequency that would closely match the equipment it is being use with (car engine).*
• **Harmonic analysis**	**AP**	*Determine a structure's response to steady, harmonic loads: Rotating machines exert steady, alternating forces on bearings and support structures causing different deflections and stresses depending on the speed of rotation.*
• **Spectrum analysis**	**AP**	*Determine a structure's response to seismic loading: How well will the structure perform and survive in an earthquake?*
• **Transient Dynamic analysis**	**A**	*To calculate a structure's response to time-varying loads: An auto fender should be able to withstand low-speed impact, but deform under higher-speed impact. A tennis racket frame should resist the impact of a tennis ball and yet flex somewhat.*
• **Random Vibration analysis**	**AP**	*Determine how a component responds to random vibrations: Spacecraft and aircraft components must withstand random loading of varying frequencies for a sustained time period.*

Class: DS = ANSYS DesignSpace AP = ANSYS Professional A = ANSYS Structural or higher

FEA Terms and Definitions

• *Mostly Derived from NAFEMS: www.nafems.org*

ACCELERATION

The second time derivative of the displacement (the first time derivative of the velocity).

ADAPTIVE FINITE ELEMENT METHOD/ADAPTIVE MESHING

An adaptive finite element solver iteratively performs finite element analysis, determines the areas of the mesh where the solution is not sufficiently accurate and refines the mesh in those areas until the solution obtains the prescribed degree of accuracy. Adaptive Meshing involves automatically improving the mesh where necessary to meet specified convergence criteria.

ALGEBRAIC EIGENVALUE PROBLEM

The eigenvalue problem when written in the form of stiffness times mode shape minus eigenvalue times mass times mode shape is equal to zero. It is the form that arises naturally from a discrete parameter model in free vibration.

ASPECT RATIO

The ratio of the longest to shortest side lengths on an element.

ASSEMBLY

Geometric: Two or more parts mated together.

FEA: The process of assembling the element matrices together to form the global matrix. Typically element stiffness matrices are assembled to form the complete stiffness matrix of the structure.

AUTOMATIC MESH GENERATION

The process of generating a mesh of elements over the volume that is being analyzed. There are two forms of automatic mesh generation: Free Meshing - Where the mesh has no structure to it. Free meshing generally uses triangular and tetrahedral elements. Mapped Meshing - Where large regions, if not all, of the volume is covered with regular meshes. This can use any form of element. Free meshing can be used to fill any shape. Mapped meshing can only be used on some shapes without elements being excessively distorted.

AXISYMMETRY

If a shape can be defined by rotating a cross-section about a line (e.g. a cone) then it is said to be axisymmetric. This can be used to simplify the analysis of the system. Such models are sometimes called two and a half dimensional since a 2D cross-section represents a 3D body.

BARLOW POINTS

The set of Gauss integration points that give the best estimates of the stress for an element. For triangles and tetrahedra these are the full Gauss integration points. For quadrilateral and brick elements they are the reduced Gauss points.

BASIS SPACE

When an element is being constructed it is derived from a simple regular shape in non-dimensional coordinates. The coordinates used to define the simple shape form the basis space. In its basis space a general quadrilateral is a 2x2 square and a general triangle is an isosoles triangle with unit side lengths.

BEAM ELEMENT

A line element that has both translational and rotational degrees of freedom. It represents both membrane and bending actions.

BENDING

Bending behavior is where the strains vary linearly from the centerline of a beam or center surface of a plate or shell. There is zero strain on the centerline for pure bending. Plane sections are assumed to remain plane. If the stresses are constant normal to the centerline then this is called membrane behavior.

BENDING STRESS

A compressive and/or tensile stress resulting from the application of a nonaxial force to a structural member.

BODY FORCE VECTOR

Mechanical loadings within the interior of the volume, typically inertia loadings in a stiffness analysis.

BOUNDARY ELEMENT BOUNDARY INTEGRAL

A method of solving differential equations by taking exact solutions to the field equations loaded by a point source and then finding the strengths of sources distributed around the boundary of the body required to satisfy the boundary conditions on the body.

BUBBLE FUNCTIONS

Element shape functions that are zero along the edges of the element. They are non-zero within the interior of the element.

BUCKLING (SNAP THROUGH)

The situation where the elastic stiffness of the structure is cancelled by the effects of compressive stress within the structure. If the effect of this causes the structure to suddenly displace a large amount in a direction normal to the load direction then it is classical bifurcation buckling. If there is a sudden large movement in the direction of the loading it is snap through buckling.

CENTRAL DIFFERENCE METHOD

A method for numerically integrating second order dynamic equations of motion. It is widely used as a technique for solving non-linear dynamic problems.

CHARACTERISTIC VALUE

Same as the eigenvalue.

CHARACTERISTIC VECTOR

Same as the eigenvector.

CHOLESKY FACTORISATION (SKYLINE)

A method of solving a set of simultaneous equations that is especially well suited to the finite element method. It is sometimes called a skyline solution. Choose to optimize the profile of the matrix if a renumbering scheme is used.

COEFFICIENT OF VISCOUS DAMPING

The system parameter relating force to velocity.

COLUMN VECTOR (COLUMN MATRIX)

An nx1 matrix written as a vertical string of numbers. It is the transpose of a row vector.

COMPATIBILITY EQUATIONS

Compatibility is satisfied if a field variable, typically the structural displacement, which is continuous before loading is continuous after loading. For linear problems the equations of compatibility must be satisfied. Nonlinearity in or non-satisfaction of, the compatibility equations leads to cracks and gaps in the structure. For finite element solutions compatibility of displacement is maintained within the element and across element boundaries for the most reliable forms of solution.

COMPATIBILITY OF STRAINS

Compatibility of strain is satisfied if strains that are continuous before loading are continuous after. Admin

COMPLETE DISPLACEMENT FIELD

When the functions interpolating the field variable (typically the displacements) form a complete n'th order polynomial in all directions.

COMPLEX EIGENVALUES

The eigenvectors of a damped system. For proportionally damped systems they are the same as the undamped eigenvectors. For non-proportionally damped systems with damping in all modes less than critical they are complex numbers and occur as complex conjugate pairs.

COMPLEX EIGENVECTORS

The eigenvalues of any damped system. If the damping is less than critical they will occur as complex conjugate pairs even for proportionally damped systems. The real part of the complex eigenvalue is a measure of the damping in the mode and should always be negative. The imaginary part is a measure of the resonant frequency.

COMPOSITE MATERIAL

A material that is made up of discrete components, typically a carbon-epoxy composite material or a glass-fiber material. Layered material and foam materials are also forms of composite materials.

COMPUTATIONAL FLUID DYNAMICS (CFD)

A computer-based numerical study of turbulent fluid flow using approximate methods such as the finite element method, the fine difference method, the boundary element method, the finite volume methods, and so on.

CONDENSATION STATIC CONDENSATION MODAL CONDENSATION

The reduction of the size of a problem by eliminating (condensing out) some degrees of freedom. For static condensation the elimination process is based upon static considerations alone. In more general condensation it can include other effects, typically model condensation includes both static and dynamic effects.

CONDITION NUMBER

The ratio of the highest eigenvalue to the lowest eigenvalue of a matrix. The exponent of this number gives a measure of the number of digits required in the computation to maintain numerical accuracy. The higher the condition number the more chance of numerical error and the slower the rate of convergence for iterative solutions.

CONDITIONAL STABILITY UNCONDITIONAL STABILITY

Any scheme for numerically integrating dynamic equations of motion in a step-by-step form is conditionally stable if there is a maximum time step value that can be used. It is unconditionally stable (but not necessarily accurate) if any length of time step can be used.

CONGRUENT TRANSFORMATION

A transformation of the coordinate system of the problem that preserves the symmetry of the system matrices.

CONJUGATE GRADIENT METHOD

A method for solving simultaneous equations iteratively. It is closely related to the Lanczos method for finding the first few eigenvalues and eigenvectors of a set of equations.

CONSISTENT DISPLACEMENTS AND FORCES

The displacements and forces act at the same point and in the same direction so that the sum of their products give a work quantity. If consistent displacements and forces are used the resulting stiffness and mass matrices are symmetric.

CONSTANT STRAIN CONSTANT STRESS

For structural analysis an element must be able to reproduce a state of constant stress and strain under a suitable loading to ensure that it will converge to the correct solution. This is tested for using the patch test.

CONSTITUTIVE RELATIONSHIPS

The equations defining the material behavior for an infinitesimal volume of material. For structures these are the stress-strain laws and include Hookes law for elasticity and the Prandle-Reuss equations for plasticity.

CONSTRAINT EQUATIONS (MULTI POINT CONSTRAINTS)

If one group of variables can be defined in terms of another group then the relationship between the two are constraint equations. Typically the displacements on the face of an element can be constrained to remain plane but the plane itself can move.

CONSTRAINTS

Known values of, or relationships between, the displacements in the coordinate system.

CONTACT PROBLEMS

A contact problem occurs when two bodies that are originally apart can come together, or two bodies that are originally connected can separate.

CONTINUOUS MASS MODELS

The system mass is distributed between the degrees of freedom. The mass matrix is not diagonal.

CONTINUOUS MODELS

The model is defined in terms of partial differential equations rather than in finite degree of freedom matrix form.

CONTOUR PLOTTING

A graphical representation of the variation of a field variable over a surface, such as stress, displacement, or temperature. A contour line is a line of constant value for the variable. A contour band is an area of a single color for values of the variable within two limit values.

CONVERGENCE REQUIREMENTS

For a structural finite element to converge as the mesh is refined it must be able to represent a state of constant stress and strain free rigid body movements exactly. There are equivalent requirements for other problem types.

CONVOLUTION INTEGRAL (DUHAMEL INTEGRAL)

The integral relating the dynamic displacement response of the structure at any time t to the forces applied before this time.

COORDINATE SYSTEM

The set of displacements used to define the degrees of freedom of the system.

CORRESPONDING FORCES AND DISPLACEMENTS

A force and a displacement are said to correspond if they act at the same point and in the same direction. Forces and translational displacements can correspond as can moments and rotations. Corresponding forces and displacements can be multiplied together to give a work quantity. Using corresponding forces and displacements will always lead to a symmetric stiffness matrix.

CRACK ELEMENT (CRACK TIP ELEMENT)

An element that includes special functions to model the stress field at the tip of a crack. This is commonly achieved by using quadratic elements with mid side nodes at the quarter chord points.

CRACK PROPAGATION (FRACTURE MECHANICS)

The process by which a crack can propagate through a structure. It is commonly assumed that a crack initiates when a critical value of stress or strain is reached and it propagates if it can release more than a critical amount of energy by the crack opening.

CRANK-NICHOLSON SCHEME

A method for numerically integrating first order dynamic equations of motion. It is widely used as a technique for solving thermal transient problems.

CRITICAL ENERGY RELEASE

This is a material property defining the minimum energy that a propagating crack must release in order for it to propagate. Three critical energies, or modes of crack propagation, have been identified. Mode 1 is the two surfaces of the crack moving apart. Mode 2 is where the two surfaces slide from front to back. Mode 3 is where the to surfaces slide sideways.

CRITICALLY DAMPED SYSTEM CRITICAL DAMPING

The dividing line between under damped and over damped systems where the equation of motion has a damping value that is equal to the critical damping.

CYCLIC SYMMETRY

A generalization of axisymmetry. The structure is composed of a series of identical sectors that are arranged circumferentially to form a ring. A turbine disc with blades attached is a typical example.

DAMPED EIGENVALUES

Same as complex eigenvalues.

DAMPED EIGENVECTORS

Same as complex eigenvectors.

DAMPED NATURAL FREQUENCY

The frequency at which the damped system vibrates naturally when only an initial disturbance is applied.

DAMPING

Any mechanism that dissipates energy in a vibrating system.

DAMPING FACTOR (DECAY FACTOR)

The damping factor is the ratio of the actual damping to the critical damping. It is often specified as a percentage. If the damping factor is less than one then the system can undergo free vibrations. The free vibrations will decay to zero with time. If the damping factor is greater than one then the decay is exponential and no vibrations occur. For most structures the damping factor is very small.

DEGENERATE ELEMENTS

Elements that are defined as one shape in the basis space but they are a simpler shape in the real space. A quadrilateral can degenerate into a triangle. A brick element can degenerate into a wedge, a pyramid or a tetrahedron. Degenerate elements should be avoided in practice.

DEGREES OF FREEDOM

The number of equations of equilibrium for the system. In dynamics, the number of displacement quantities which must be considered in order to represent the effects of all of the significant inertia forces.

Degrees of freedom define the ability of a given node to move in any direction in space. There are six types of DOF for any given node:

- 3 possible translations (one each in the X,Y and Z directions) and
- 3 possible rotations (one rotation about each of the X,Y, and X axes).

DOF are defined and restricted by the elements and constraints associated with each node.

DET(J) DET J

The Jacobian matrix is used to relate derivatives in the basis space to the real space. The determinant of the Jacobian - det(j) - is a measure of the distortion of the element when mapping from the basis to the real space.

DETERMINISTIC ANALYSIS

The applied loading is a known function of time.

DEVIATORIC STRESS STRESS DEVIATORS

A measure of stress where the hydrostatic stress has been subtracted from the actual stress. Material failures that are flow failures (plasticity and creep) fail independently of the hydrostatic stress. The failure is a function of the deviatoric stress.

DIAGONAL DECAY

When a matrix is factorized into a triangular form the ratio of a diagonal term in the factorized matrix to the corresponding term in the original matrix decreases in size as one moves down the diagonal. If the ratio goes to zero the matrix is singular and if it is negative the matrix is not positive definite. The diagonal decay can be used as an approximate estimate of the condition number of the matrix.

DIAGONAL GENERALIZED MATRIX

The eigenvectors of a system can be used to define a coordinate transformation such that, in these generalized coordinates the coefficient matrices (typically mass and stiffness) are diagonal

DIE-AWAY LENGTH

If there is a stress concentration in a structure the high stress will reduce rapidly with distance from the peak value. The distance over which it drops to some small value is called the die-away length. A fine mesh is required over this die-away length for accurate stress results.

DIRECT INTEGRATION

The name for various techniques for numerically integrating equations of motion. These are either implicit or explicit methods and include central difference, Crank-Nicholson, Runge-Kutta, Newmark beta and Wilson theta.

DIRECTION COSINES

The cosines of the angles a vector makes with the global x,y,z axes.

DISCRETE PARAMETER MODELS (DISCRETISED APPROACH)

The model is defined in terms of an ordinary differential equation and the system has a finite number of degrees of freedom.

DISCRETIZATION

The process of dividing geometry into smaller pieces (finite elements) to prepare for analysis, i.e. Meshing.

DISPLACEMENT METHOD (DISPLACEMENT SOLUTION)

A form of discrete parameter model where the displacements of the system are the basic unknowns.

DISPLACEMENT

The distance, translational and rotational, that a node travels from its initial position to its post-analysis position. The total displacement is represented by components in each of the 3 translational directions and the 3 rotational directions.

DISPLACEMENT PLOTS

Plots showing the deformed shape of the structure. For linear small deflection problems the displacements are usually multiplied by a magnifying factor before plotting the deformed shape.

DISPLACEMENT VECTOR

The nodal displacements written as a column vector.

DISSIMILAR SHAPE FUNCTIONS INCOMPATIBLE SHAPE FUNCTIONS

If two connecting elements have different shape functions along the connection line they are said to be incompatible. This should be avoided since convergence to the correct solution cannot be guarantied.

DISTORTION ELEMENT DISTORTION

Elements are defined as simple shapes in the basis space, quadrilaterals are square, triangles are isosoles triangles. If they are not this shape in the real space they are said to be distorted. Too much distortion can lead to errors in the solution

DRUCKER-PRAGER EQUIVALENT STRESSES

An equivalent stress measure for friction materials (typically sand). The effect of hydrostatic stress is included in the equivalent stress.

DYNAMIC ANALYSIS

An analysis that includes the effect of the variables changing with time as well as space.

DYNAMIC FLEXIBILITY MATRIX

The factor relating the steady state displacement response of a system to a sinusoidal force input. It is the same as the receptance.

DYNAMIC MODELLING

A modeling process where consideration as to time effects in addition to spatial effects are included. A dynamic model can be the same as a static model or it can differ significantly depending upon the nature of the problem.

DYNAMIC RESPONSE

The time dependent response of a dynamic system in terms of its displacement, velocity or acceleration at any given point of the system.

DYNAMIC STIFFNESS MATRIX

If the structure is vibrating steadily at a frequency w then the dynamic stiffness is (K+iwC-w2M) It is the inverse of the dynamic flexibility matrix.

DYNAMIC STRESSES

Stresses that vary with time and space.

DYNAMIC SUBSTRUCTURING

Special forms of substructuring used within a dynamic analysis. Dynamic substructuring is always approximate and causes some loss of accuracy in the dynamic solution.

EIGENVALUE PROBLEM

Problems that require calculation of eigenvalues and eigenvectors for their solution. Typically solving free vibration problems or finding buckling loads.

EIGENVALUES LATENT ROOTS CHARACTERISTIC VALUES

The roots of the characteristic equation of the system. If a system has n equations of motion then it has n eigenvalues. The square root of the eigenvalues are the resonant frequencies. These are the frequencies that the structure will vibrate at if given some initial disturbance with no other forcing. There are other problems that require the solution of the eigenvalue problem, the buckling loads of a structure are eigenvalues. Latent roots and characteristic values are synonyms for eigenvalues.

EIGENVECTORS LATENT VECTORS NORMAL MODES

The displacement shape that corresponds to the eigenvalues. If the structure is excited at a resonant frequency then the shape that it adopts is the mode shape corresponding to the eigenvalue. Latent vectors and normal modes are the same as eigenvectors.

ELASTIC FOUNDATION

If a structure is sitting on a flexible foundation the supports are treated as a continuous elastic foundation. The elastic foundation can have a significant effect upon the structural response.

ELASTIC STIFFNESS

If the relationship between loads and displacements is linear then the problem is elastic. For a multi-degree of freedom system the forces and displacements are related by the elastic stiffness matrix.

ELECTRIC FIELDS

Electro-magnetic and electro-static problems form electric field problems.

ELEMENT

In the finite element method the geometry is divided up into elements, much like basic building blocks. Each element has nodes associated with it. The behavior of the element is defined in terms of the freedoms at the nodes.

ELEMENT ASSEMBLY

Individual element matrices have to be assembled into the complete stiffness matrix. This is basically a process of summing the element matrices. This summation has to be of the correct form. For the stiffness method the summation is based upon the fact that element displacements at common nodes must be the same.

ELEMENT STRAINS ELEMENT STRESSES

Stresses and strains within elements are usually defined at the Gauss points (ideally at the Barlow points) and the node points. The most accurate estimates are at the reduced Gauss points (more specifically the Barlow points). Stresses and strains are usually calculated here and extrapolated to the node points.

ENERGY METHODS HAMILTONS PRINCIPLE

Methods for defining equations of equilibrium and compatibility through consideration of possible variations of the energies of the system. The general form is Hamiltons principle and sub-sets of this are the principle of virtual work including the principle of virtual displacements (PVD) and the principle of virtual forces (PVF).

ENGINEERING NORMALIZATION MATHEMATICAL NORMALIZATION

Each eigenvector (mode shape or normal mode) can be multiplied by an arbitrary constant and still satisfy the eigenvalue equation. Various methods of scaling the eigenvector are used Engineering normalization - The vector is scaled so that the largest absolute value of any term in the eigenvector is unity. This is useful for inspecting printed tables of eigenvectors. Mathematical normalization - The vector is scaled so that the diagonal modal mass matrix is the unit matrix. The diagonal modal stiffness matrix is the system eigenvalues. This is useful for response calculations.

EQUILIBRIUM EQUATIONS

Internal forces and external forces must balance. At the infinitesimal level the stresses and the body forces must balance. The equations of equilibrium define these force balance conditions.

EQUILIBRIUM FINITE ELEMENTS

Most of the current finite elements used for structural analysis are defined by assuming displacement variations over the element. An alternative approach assumes the stress variation over the element. This leads to equilibrium finite elements.

EQUIVALENT MATERIAL PROPERTIES

Equivalent material properties are defined where real material properties are smeared over the volume of the element. Typically, for composite materials the discrete fiber and matrix material properties are smeared to give average equivalent material properties.

EQUIVALENT STRESS

A three dimensional solid has six stress components. If material properties have been found experimentally by a uniaxial stress test then the real stress system is related to this by combining the six stress components to a single equivalent stress. There are various forms of equivalent stress for different situations. Common ones are Tresca, Von-Mises, Mohr-Coulomb and Drucker-Prager.

ERGODIC PROCESS

A random process where any one-sample record has the same characteristics as any other record.

EULERIAN METHOD LAGRANGIAN METHOD

For non-linear large deflection problems the equations can be defined in various ways. If the material is flowing though a fixed grid the equations are defined in Eulerian coordinates. Here the volume of the element is constant but the mass in the element can change. If the grid moves with the body then the equations are defined in Lagrangian coordinates. Here the mass in the element is fixed but the volume changes.

EXACT SOLUTIONS

Solutions that satisfy the differential equations and the associated boundary conditions exactly. There are very few such solutions and they are for relatively simple geometries and loadings.

EXPLICIT METHODS IMPLICIT METHODS

These are methods for integrating equations of motion. Explicit methods can deal with highly non-linear systems but need small steps. Implicit methods can deal with mildly non-linear problems but with large steps.

EXTRAPOLATION INTERPOLATION

The process of estimating a value of a variable from a tabulated set of values. For interpolation values inside the table are estimated. For extrapolation values outside the table are estimated. Interpolation is generally accurate and extrapolation is only accurate for values slightly outside the table. It becomes very inaccurate for other cases.

FACETED GEOMETRY

If a curved line or surface is modeled by straight lines or flat surfaces then the modeling is said to produce a faceted geometry.

FAST FOURIER TRANSFORM

A method for calculating Fourier transforms that is computationally very efficient.

FIELD PROBLEMS

Problems that can be defined by a set of partial differential equations are field problems. Any such problem can be solved approximately by the finite element method.

FINITE DIFFERENCES

A numerical method for solving partial differential equations by expressing them in a difference form rather than an integral form. Finite difference methods are very similar to finite element methods and in some cases are identical.

FINITE ELEMENT MODELING (FEM)

The process of setting up a model for analysis, typically involving graphical generation of the model geometry, meshing it into finite elements, defining material properties, and applying loads and boundary conditions.

FINITE VOLUME METHODS

A technique related to the finite element method. The equations are integrated approximately using the weighted residual method, but a different form of weighting function is used from that in the finite element method. For the finite element method the Galerkin form of the weighted residual method is used.

FIXED BOUNDARY CONDITIONS

All degrees of freedom are restrained for this condition. The nodes on the fixed boundary can not move: translation or rotation.

FLEXIBILITY MATRIX FORCE METHOD

The conventional form of the finite element treats the displacements as unknowns, which leads to a stiffness matrix form. Alternative methods treating the stresses (internal forces) as unknowns lead to force methods with an associated flexibility matrix. The inverse of the stiffness matrix is the flexibility matrix.

FORCED RESPONSE

The dynamic motion results from a time varying forcing function.

FORCING FUNCTIONS

The dynamic forces that are applied to the system.

FOURIER EXPANSIONS FOURIER SERIES

Functions that repeat themselves in a regular manner can be expanded in terms of a Fourier series.

FOURIER TRANSFORM

A method for finding the frequency content of a time varying signal. If the signal is periodic it gives the same result as the Fourier series.

FOURIER TRANSFORM PAIR

The Fourier transform and its inverse which, together, allow the complete system to be transformed freely in either direction between the time domain and the frequency domain.

FRAMEWORK ANALYSIS

If a structure is idealized as a series interconnected line elements then this forms a framework analysis model. If the connections between the line elements are pins then it is a pin-jointed framework analysis. If the joints are rigid then the lines must be beam elements.

FREE VIBRATION

The dynamic motion which results from specified initial conditions. The forcing function is zero.

FREQUENCY DOMAIN

The structures forcing function and the consequent response is defined in terms of their frequency content. The inverse Fourier transform of the frequency domain gives the corresponding quantity in the time domain.

FRONTAL SOLUTION WAVEFRONT SOLUTION

A form of solving the finite element equations using Gauss elimination that is very efficient for the finite element form of equations.

GAP ELEMENT CONTACT ELEMENT

These are special forms of non-linear element that have a very high stiffness in compression and a low stiffness in tension. They are used to model contact conditions between surfaces. Most of these elements also contain a model for sliding friction between the contacting surfaces. Some gap elements are just line springs between points and others are more general forms of quadrilateral or brick element elements. The line spring elements should only be used in meshes of first order finite elements.

GAUSS POINT EXTRAPOLATION GAUSS POINT STRESSES

Stresses calculated internally within the element at the Gauss integration points are called the Gauss point stresses. These stresses are usually more accurate at these points than the nodal points.

GAUSS POINTS GAUSS WEIGHTS

The Gauss points are the sample points used within the elements for the numerical integration of the matrices and loadings. They are also the points at which the stresses can be recovered. The Gauss weights are associated factors used in the numerical integration process. They represent the volume of influence of the Gauss points. The positions of the Gauss points, together with the associated Gauss weights, are available in tables for integrations of polynomials of various orders.

GAUSSIAN ELIMINATION

A form of solving a large set of simultaneous equations. Within most finite element systems a form of Gaussian elimination forms the basic solution process.

GAUSSIAN INTEGRATION GAUSSIAN QUADRATURE

A form of numerically integrating functions that is especially efficient for integrating polynomials. The functions are evaluated at the Gauss points, multiplied by the Gauss weights and summed to give the integral.

GENERALIZED COORDINATES

A set of linearly independent displacement coordinates which are consistent with the constraints and are just sufficient to describe any arbitrary configuration of the system. Generalized coordinates are usually patterns of displacements, typically the system eigenvectors.

GENERALIZED MASS

The mass associated with a generalized displacement.

GENERALIZED STIFFNESS

The stiffness associated with a generalized displacement.

GEOMETRIC PROPERTIES

Various shape dependent properties of real structures, such as thickness, cross sectional area, sectional moments of inertia, centroid location and others that are applied as properties of finite elements.

GEOMETRIC STIFFNESS STRESS STIFFNESS

The component of the stiffness matrix that arises from the rotation of the internal stresses in a large deflection problem. This stiffness is positive for tensile stresses and negative for compressive stresses. If the compressive stresses are sufficiently high then the structure will buckle when the geometric stiffness cancels the elastic stiffness.

GEOMETRICAL ERRORS

Errors in the geometrical representation of the model. These generally arise from the approximations inherent in the finite element approximation.

GLOBAL STIFFNESS MATRIX

The assembled stiffness matrix of the complete structure.

GROSS DEFORMATIONS

Deformations sufficiently high to make it necessary to include their effect in the solution process. The problem requires a large deflection non-linear analysis.

GUARD VECTORS

The subspace iteration (simultaneous vector iteration) method uses extra guard vectors in addition to the number of vectors requested by the user. These guard the desired vectors from being contaminated by the higher mode vectors and speed up convergence.

GUYAN REDUCTION METHOD

A method for reducing the number of degrees of freedom in a dynamic analysis. It is based upon a static approximation and always introduces some error in the computed dynamic solution. The error depends upon the choice of master freedoms.

GYROSCOPIC FORCES

Forces arising from Coriolis acceleration. These can destabilize a dynamic response and cause whirling.

HARDENING STRUCTURE

A structure where the stiffness increases with load.

HARMONIC LOADING

A dynamic loading that is periodic and can be represented by a Fourier series.

HEAT CONDUCTION

The analysis of the steady state heat flow within solids and fluids. The equilibrium balance between internal and external heat flows.

HERMITIAN SHAPE FUNCTIONS

Shape functions that provide both variable and variable first derivative continuity (displacement and slope continuity in structural terms) across element boundaries.

HEXAHEDRON ELEMENTS

Type of 3D element which has six quadrilateral faces.

HIDDEN LINE REMOVAL

Graphical plots of models where non-visible mesh lines are not plotted.

HIGH ASPECT RATIO LOW ASPECT RATIO

The ratio of the longest side length of a body to the shortest is termed its aspect ratio. Generally bodies with high aspect ratios (long and thin) are more ill conditioned for numerical solution than bodies with an aspect ratio of one.

HOLONOMIC CONSTRAINTS

Constraints that can be defined for any magnitude of displacement.

HOOKES LAW

The material property equations relating stress to strain for linear elasticity. They involve the material properties of Young's modulus and Poisson ratio.

HOURGLASS MODE

Zero energy modes of low order quadrilateral and brick elements that arise from using reduced integration. These modes can propagate through the complete body.

H-CONVERGENCE

Convergence towards a more accurate solution by subdividing the elements into a number of smaller elements. This approach is referred to as H-convergence because of improved discretization due to reduced element size.

H-METHOD

A finite element method which requires an increasing number of elements to improve the solution.

H-REFINEMENT P-REFINEMENT

Making the mesh finer over parts or all of the body is termed h-refinement. Making the element order higher is termed p-refinement.

HYBRID ELEMENTS

Elements that use stress interpolation within their volume and displacement interpolation around their boundary.

HYDROSTATIC STRESS

The stress arising from a uniform pressure load on a cube of material. It is the average value of the direct stress components at any point in the body.

HYSTERETIC DAMPING

A damping model representing internal material loss damping. The energy loss per unit cycle is independent of frequency. It is only valid for harmonic response.

ILL-CONDITIONING ERRORS

Numerical (rounding) errors that arise when using ill-conditioned equations.

ILL-CONDITIONING ILL-CONDITIONED EQUATIONS

Equations that are sensitive to rounding errors in a numerical operation. The numerical operation must also be defined. Equations can be ill conditioned for solving simultaneous equations but not for finding eigenvalues.

IMPULSE RESPONSE FUNCTION

The response of the system to an applied impulse.

IMPULSE RESPONSE MATRIX

The matrix of all system responses to all possible impulses. It is always symmetric for linear systems. It is the inverse Fourier transform of the dynamic flexibility matrix.

INCREMENTAL SOLUTION

A solutions process that involves applying the loading in small increments and finding the equilibrium conditions at the end of each step. Such solutions are generally used for solving non-linear problems.

INELASTIC MATERIAL BEHAVIOR

A material behavior where residual stresses or strains can remain in the body after a loading cycle, typically plasticity and creep.

INERTANCE (ACCELERANCE)

The ratio of the steady state acceleration response to the value of the forcing function for a sinusoidal excitation.

INERTIA FORCE

The force that is equal to the mass times the acceleration.

INITIAL BUCKLING

The load at which a structure first buckles.

INITIAL STRAINS

The components of the strains that are non-elastic. Typically thermal strain and plastic strain.

INTEGRATION BY PARTS

A method of integrating a function where high order derivative terms are partially integrated to reduce their order.

INTERPOLATION FUNCTIONS SHAPE FUNCTIONS

The polynomial functions used to define the form of interpolation within an element. When these are expressed as interpolations associated with each node they become the element shape functions.

ISOPARAMETRIC ELEMENT

Elements that use the same shape functions (interpolations) to define the geometry as were used to define the displacements. If these elements satisfy the convergence requirements of constant stress representation and strain free rigid body motions for one geometry then it will satisfy the conditions for any geometry.

ISOTROPIC MATERIAL

Materials where the material properties are independent of the co-ordinate system.

JACOBI METHOD

A method for finding eigenvalues and eigenvectors of a symmetric matrix.

JACOBIAN MATRIX

A square matrix relating derivatives of a variable in one coordinate system to the derivatives of the same variable in a second coordinate system. It arises when the chain rule for differentiation is written in matrix form.

J-INTEGRAL METHODS

A method for finding the stress intensity factor for fracture mechanics problems.

JOINTS

The interconnections between components. Joints can be difficult to model in finite element terms but they can significantly affect dynamic behavior.

KINEMATIC BOUNDARY CONDITIONS

The necessary displacement boundary conditions for a structural analysis. These are the essential boundary conditions in a finite element analysis.

KINEMATICALLY EQUIVALENT FORCES (LOADS)

A method for finding equivalent nodal loads when the actual load is distributed over a surface of a volume. The element shape functions are used so that the virtual work done by the equivalent loads is equal to the virtual work done by the real loads over the same virtual displacements. This gives the most accurate load representation for the finite element model. These are the non-essential stress boundary conditions in a finite element analysis.

KINEMATICALLY EQUIVALENT MASS

If the mass and stiffness are defined by the same displacement assumptions then a kinematically equivalent mass matrix is produced. This is not a diagonal (lumped) mass matrix.

KINETIC ENERGY

The energy stored in the system arising from its velocity. In some cases it can also be a function of the structural displacements.

LAGRANGE INTERPOLATION LAGRANGE SHAPE FUNCTIONS

A method of interpolation over a volume by means of simple polynomials. This is the basis of most of the shape function definitions for elements.

LAGRANGE MULTIPLIER TECHNIQUE

A method for introducing constraints into an analysis where the effects of the constraint are represented in terms of the unknown Lagrange multiplying factors.

LANCZOS METHOD

A method for finding the first few eigenvalues and eigenvectors of a set of equations. It is very well suited to the form of equations generated by the finite element method. It is closely related to the method of conjugate gradients used for solving simultaneous equations iteratively.

LEAST SQUARES FIT

Minimization of the sum of the squares of the distances between a set of sample points and a smooth surface. The finite element method gives a solution that is a least squares fit to the equilibrium equations.

LINEAR DEPENDENCE

One or more rows (columns) of a matrix are linear combinations of the other rows (columns). This means that the matrix is singular.

LINEAR ANALYSIS

Analysis in which the displacements of the structure are linear functions of the applied loads.

LINEAR SYSTEM

When the coefficients of stiffness, mass and damping are all constant then the system is linear. Superposition can be used to solve the response equation.

LOADINGS

The loads applied to a structure that result in deflections and consequent strains and stresses.

LOCAL STRESSES

Areas of stress that are significantly different from (usually higher than) the general stress level.

LOWER BOUND SOLUTION UPPER BOUND SOLUTION

The assumed displacement form of the finite element solution gives a lower bound on the maximum displacements and strain energy (i.e. these are under estimated) for a given set of forces. This is the usual form of the finite element method. The assumed stress form of the finite element solution gives an upper bound on the maximum stresses and strain energy (i.e. these are over estimated) for a given set of displacements.

LUMPED MASS MODEL

When the coefficients of the mass matrix are combined to produce a diagonal matrix. The total mass and the position of the structures center of gravity are preserved.

MASS

The constant(s) of proportionality relating the acceleration(s) to the force(s). For a discrete parameter multi degree of freedom model this is usually given as a mass matrix.

MASS ELEMENT

An element lumped at a node representing the effect of a concentrated mass in different coordinate directions.

MASS MATRIX

The matrix relating acceleration to forces in a dynamic analysis. This can often be approximated as a diagonal matrix with no significant loss of accuracy.

MASTER FREEDOMS

The freedoms chosen to control the structural response when using a Guyan reduction or sub structuring methods.

MATERIAL LOSS FACTOR

A measure of the damping inherent within a material when it is dynamically loaded.

MATERIAL PROPERTIES

The physical properties required to define the material behavior for analysis purposes. For stress analysis typical required material properties are Young's modulus, Poisson's ratio, density and coefficient of linear expansion. The material properties must have been obtained by experiment.

MATERIAL STIFFNESS MATRIX MATERIAL FLEXIBILITY MATRIX

The material stiffness matrix allows the stresses to be found from a given set of strains at a point. The material flexibility is the inverse of this, allowing the strains to be found from a given set of stresses. Both of these matrices must be symmetric and positive definite.

MATRIX DISPLACEMENT METHOD

A form (the standard form) of the finite element method where displacements are assumed over the element. This gives a lower bound solution.

MATRIX FORCE METHOD

A form of the finite element method where stresses (internal forces) are assumed over the element. This gives an upper bound solution.

MATRIX INVERSE

If matrix A times matrix B gives the unit matrix then A is the inverse of B (B is the inverse of A). A matrix has no inverse if it is singular.

MATRIX NOTATION MATRIX ALGEBRA

A form of notation for writing sets of equations in a compact manner. Matrix notation highlights the generality of various classes of problem formulation and solution. Matrix algebra can be easily programmed on a digital computer.

MATRIX PRODUCTS

Two matrices A and B can be multiplied together if A is of size (j*k) and B is of size (k*l). The resulting matrix is of size (j*l).

MATRIX TRANSPOSE

The process of interchanging rows and columns of a matrix so that the j'th column becomes the j'th row.

MEAN SQUARE CONVERGENCE

A measure of the rate of convergence of a solution process. A mean square convergence indicates a rapid rate of convergence.

MEMBRANE

Membrane behavior is where the strains are constant from the center line of a beam or center surface of a plate or shell. Plane sections are assumed to remain plane. A membrane line element only has stiffness along the line, it has zero stiffness normal to the line. A membrane plate has zero stiffness normal to the plate. This can cause zero energy (no force required) displacements in these normal directions. If the stresses vary linearly along the normal to the centerline then this is called bending behavior.

MESH DENSITY MESH REFINEMENT

The mesh density indicates the size of the elements in relation to the size of the body being analyzed. The mesh density need not be uniform all over the body There can be areas of mesh refinement (more dense meshes) in some parts of the body. Making the mesh finer is generally referred to as h-refinement. Making the element order higher is referred to as p-refinement.

MESH GENERATION ELEMENT GENERATION

The process of generating a mesh of elements over the structure. This is normally done automatically or semi-automatically.

MESH SPECIFICATION

The process of choosing and specifying a suitable mesh of elements for an analysis.

MESH SUITABILITY

The appropriate choice of element types and mesh density to give a solution to the required degree of accuracy.

MINDLIN ELEMENTS

A form of thick shell element.

MOBILITY

The ratio of the steady state velocity response to the value of the forcing function for a sinusoidal excitation.

MODAL DAMPING

The damping associated with the generalized displacements defined by the eigenvectors. Its value has no physical significance since the eigenvector contains an arbitrary normalizing factor.

MODAL MASS

The mass associated with the generalized displacements defined by the eigenvectors. Its value has no physical significance since the eigenvector contains an arbitrary normalizing factor but the ratio of modal stiffness to modal mass is always the eigenvalue.

MODAL STIFFNESS

The stiffness associated with the generalized displacements defined by the eigenvectors. Its value has no physical significance since the eigenvector contains an arbitrary normalizing factor but the ratio of modal stiffness to modal mass is always the eigenvalue.

MODAL TESTING

The experimental technique for measuring resonant frequencies (eigenvalues) and mode shapes (eigenvectors).

MODE PARTICIPATION FACTOR

The generalized force in each modal equation of a dynamic system.

MODE SHAPE

Same as the eigenvector. The mode shape often refers to the measure mode, found from a modal test.

MODELLING

The process of idealizing a system and its loading to produce a numerical (finite element) model.

MODIFIED NEWTON-RAPHSON

A form of the Newton-Raphson process for solving non-linear equations where the tangent stiffness matrix is held constant for some steps. It is suitable for mildly non-linear problems.

MOHR COULOMB EQUIVALENT STRESS

A form of equivalent stress that includes the effects of friction in granular (e.g. sand) materials.

MULTI DEGREE OF FREEDOM

The system is defined by more than one force/displacement equation.

MULTI-POINT CONSTRAINTS

Where the constraint is defined by a relationship between more than one displacement at different node points.

NATURAL FREQUENCY

The frequency at which a structure will vibrate in the absence of any external forcing. If a model has n degrees of freedom then it has n natural frequencies. The eigenvalues of a dynamic system are the squares of the natural frequencies.

NATURAL MODE

Same as the eigenvector.

NAVIER-STOKES EQUATIONS

Partial differential equations defining the unsteady viscous flow of fluids.

NEWMARK METHOD NEWMARK BETA METHOD

An implicit solution method for integrating second order equations of motion. It can be made unconditionally stable.

NEWTON COTES FORMULAE

A family of methods for numerically integrating a function.

NEWTON-RAPHSON NON-LINEAR SOLUTION

A general technique for solving non-linear equations. If the function and its derivative are known at any point then the Newton-Raphson method is second order convergent.

NODAL VALUES

The value of variables at the node points. For a structure typical possible nodal values are force, displacement, temperature, velocity, x, y, and z.

NODE NODES NODAL

The point at which one element connects to another or the point where an element meets the model boundary. Nodes allow internal loads from one element to be transferred to another element. Element behavior is defined by the response at the nodes of the elements. Nodes are always at the corners of the element, higher order elements have nodes at mid-edge or other edge positions and some elements have nodes on faces or within the element volume. The behavior of the element is defined by the variables at the node. For a stiffness matrix the variables are the structural displacement, For a heat conduction analysis the nodal variable is the temperature. Other problems have other nodal variables.

NON-CONFORMING ELEMENTS

Elements that do not satisfy compatibility either within the element or across element boundaries or both. Such elements are not generally reliable although they might give very good solutions in some circumstances.

NON-HOLONOMIC CONSTRAINTS

Constraints that can only be defined at the level of infinitesimal displacements. They cannot be integrated to give global constraints.

NON-LINEAR SYSTEM NON-LINEAR ANALYSIS

When at least one of the coefficients of stiffness, mass or damping vary with displacement or time then the system is non-linear. Superposition cannot be used to solve the problem.

NON-STATIONARY RANDOM

A force or response that is random and its statistical properties vary with time.

NON-STRUCTURAL MASS

Mass that is present in the system and will affect the dynamic response but it is not a part of the structural mass (e.g. the payload).

NORM

A scalar measure of the magnitude of a vector or a matrix.

NUMERICAL INTEGRATION

The process of finding the approximate integral of a function by numerical sampling and summing. In the finite element method the element matrices are usually formed by the Gaussian quadrature form of numerical integration.

OPTIMAL SAMPLING POINTS

The minimum number of Gauss points required to integrate an element matrix. Also the Gauss points at which the stresses are most accurate (see reduced Gauss points).

OVER DAMPED SYSTEM

A system which has an equation of motion where the damping is greater than critical. It has an exponentially decaying, non-oscillatory impulse response.

OVERSTIFF SOLUTIONS

Lower bound solutions. These are associated with the assumed displacement method.

PARAMETRIC STUDIES PILOT STUDIES

Initial studies conducted on small-simplified models to determine the important parameters in the solution of a problem. These are often used to determine the basic mesh density required.

PARTICIPATION FACTOR

The fraction of the mass that is active for a given mode with a given distribution of dynamic loads. Often this is only defined for the specific load case of inertia (seismic) loads.

PATCH TEST

A test to prove that a mesh of distorted elements can represent constant stress situations and strain free rigid body motions (i.e. the mesh convergence requirements) exactly.

PERIODIC RESPONSE FORCE

A response (force) that regularly repeats itself exactly.

PHASE ANGLE

The ratio of the in phase component of a signal to its out of phase component gives the tangent of the phase angle of the signal relative to some reference.

PLANE STRAIN PLANE STRESS

A two dimensional analysis is plane stress if the stress in the third direction is assumed zero. This is valid if the dimension of the body in this direction is very small, e.g. a thin plate. A two dimensional analysis is plane strain if the strain in the third direction is assumed zero. This is valid if the dimension of the body in this direction is very large, e.g. a cross-sectional slice of a long body.

PLATE BENDING ELEMENTS

Two-dimensional shell elements where the in plane behavior of the element is ignored. Only the out of plane bending is considered.

POISSONS RATIO

The material property in Hookes law relating strain in one direction arising from a stress in a perpendicular direction to this.

POST ANALYSIS CHECKS

Checks that can be made on the results after the analysis. For a stress analysis these could include how well stress free boundary conditions have been satisfied or how continuous stresses are across elements.

POST-PROCESSING

The interrogation of the results after the analysis phase. This is usually done with a combination of graphics and numerics.

POTENTIAL ENERGY

The energy associated with the static behavior of a system. For a structure this is the strain energy.

POTENTIAL FLOW

Fluid flow problems where the flow can be represented by a scalar potential function.

POWER METHOD

A method for finding the lowest or the highest eigenvalue of a system.

PRANDTL-REUSS EQUATIONS

The equations relating an increment of stress to an increment of plastic strain for a metal undergoing plastic flow.

PREPROCESSING

The process of preparing finite element input data involving model creation, mesh generation, material definition, and load and boundary condition application.

PRIMARY COMPONENT

Those parts of the structure that are of direct interest for the analysis. Other parts are secondary components.

PRINCIPAL CURVATURE

The maximum and minimum radii of curvature at a point.

PRINCIPAL STRESSES

The maximum direct stress values at a point. They are the eigenvalues of the stress tensor.

PROFILE

The profile of a symmetric matrix is the sum of the number of terms in the lower (or upper) triangle of the matrix ignoring the leading zeros in each row. Embedded zeros are included in the count. It gives a measure of the work required to factorize the matrix when using the Cholesky solution. Node renumbering minimizes it.

PROPORTIONAL DAMPING

A damping matrix that is a linear combination of the mass and stiffness matrices. The eigenvectors of a proportionally damped system are identical to those of the undamped system.

P-METHOD

A method of finite element analysis that uses P-convergence to iteratively minimize the error of analysis.

QR METHOD

A technique for finding eigenvalues. This is currently the most stable method for finding eigenvalues but it is restricted in the size of problem that it can solve.

RANDOM VIBRATIONS

The applied loading is only known in terms of its statistical properties. The loading is non-deterministic in that its value is not known exactly at any time but its mean, mean square, variance and other statistical quantities are known.

RANK DEFICIENCY

A measure of how singular a matrix is.

RAYLEIGH DAMPING

Damping that is proportional to a linear combination of the stiffness and mass. This assumption has no physical basis but it is mathematically convenient to approximate low damping in this way when exact damping values are not known.

RAYLELGH QUOTIENT

The ratio of stiffness times displacement squared (2*strain energy) to mass times displacement squared. The minimum values of the Rayleigh quotient are the eigenvalues.

REACTION FORCES

The forces generated at support points when a structure is loaded.

REFERENCE TEMPERATURE

The reference temperature defines the temperature at which strain in the design does not result from thermal expansion or contraction. For many situations, reference temperature is adequately defined as room temperature. Define reference temperature in the properties of an environment.

RECEPTANCE

The ratio of the steady state displacement response to the value of the forcing function for a sinusoidal excitation. It is the same as the dynamic flexibility.

REDUCED INTEGRATION

If an element requires an l*m*n Gauss rule to integrate the element matrix exactly then (l-1)*(m-1)*(n-1) is the reduced integration rule. For many elements the stresses are most accurate at the reduced integration points. For some elements the matrices are best evaluated by use of the reduced integration points. Use of reduced integration for integrating the elements can lead to zero energy and hour glassing modes.

RESPONSE SPECTRUM METHOD

A method for characterizing a dynamic transient forcing function and the associated solution technique. It is used for seismic and shock type loads.

RESTARTS CHECKPOINTS

The process whereby an analysis can be stopped part way through and the analysis restarted at a later time.

RIGID BODY DEFORMATIONS

A non-zero displacement pattern that has zero strain energy associate with it.

RIGID BODY DISPLACEMENT

A non-zero displacement pattern that has zero strain energy associate with it.

RIGID BODY MODES

If a displaced shape does not give rise to any strain energy in the structure then this a rigid body mode. A general three-dimensional unsupported structure has 6 rigid body modes, 3 translation and 3 rotation.

RIGID LINKS RIGID OFFSETS

This is a connection between two non-coincident nodes assuming that the connection is infinitely stiff. This allows the degrees of freedom at one of the nodes (the slave node) to be deleted from the system. It is a form of multi-point constraint.

ROUNDOFF ERROR

Computers have a fixed wordlength and hence only hold numbers to a certain number of significant figures. If two close numbers are subtracted one from another then the result loses the first set of significant figures and hence loses accuracy. This is round off error.

ROW VECTOR ROW MATRIX

A 1xn matrix written as a horizontal string of numbers. It is the transpose of a column vector.

SCALARS VECTORS

Quantities that have no direction associated with them, e.g. temperatures. Scalar problems only have one degree of freedom at a node. Vector quantities have a direction associated with them, e.g. displacements. Vector problems have more than one degree of freedom at a node.

SECANT STIFFNESS

The stiffness defined by the slope of the line from the origin to the current point of interest on a load/deflection curve.

SECONDARY COMPONENTS

Components of a structure not of direct interest but they may have some influence of the behavior of the part of the structure that is of interest (the primary component) and have to be included in the analysis in some approximate form.

SEEPAGE FLOW

Flows in porous materials

SEISMIC ANALYSIS

The calculation of the dynamic displacement and stress response arising from earthquake excitations.

SELECTED REDUCED INTEGRATION

A form of Gaussian quadrature where different sets of Gauss points are used for different strain components.

SELF ADJOINT EQUATIONS

A form of matrix products that preserves symmetry of equations. The product A*B*A(transpose) is self-adjoint if the matrix B is symmetric. The result of the product will be symmetric for any form of A that is of a size compatible with B. This form of equation occurs regularly within the finite element method. Typically it means that for a structural analysis the stiffness (and mass) matrices for any element or element assembly will be symmetric.

SELF EQUILIBRATING LOADS

A load set is self-equilibrating if all of its resultants are zero. Both translation and moment resultants are zero.

SEMI-LOOF ELEMENT

A form of thick shell element.

SHAKEDOWN

If a structure is loaded cyclically and initially undergoes some plastic deformation then it is said to shakedown if the behavior is entirely elastic after a small number of load cycles.

SIMPSONS RULE

A method for numerically integrating a function.

SIMULTANEOUS VECTOR ITERATION

A method for finding the first few eigenvalues and eigenvectors of a finite element system. This is also known as subspace vector iteration.

SINGLE DEGREE OF FREEDOM

The system is defined by a single force/displacement equation.

SINGLE POINT CONSTRAINT

Where the constraint is unique to a single node point.

SINGULAR MATRIX

A square matrix that cannot be inverted.

SKEW DISTORTION (ANGULAR DISTORTION)

A measure of the angular distortion arising between two vectors that are at right angles in the basis space when these are mapped to the real coordinate space. If this angle approaches zero the element becomes ill- conditioned.

SOLID ELEMENTS

Three dimensional continuum elements.

SOLUTION DIAGNOSTICS

Messages that are generated as the finite element solution progresses. These should always be checked for relevance but the are often only provided for information purposes

SOLUTION EFFICIENCY

A comparative measure between two solutions of a given problem defining which is the ‘best’. The measures can include accuracy, time of solution, memory requirements and disc storage space.

SPARSE MATRIX METHODS

Solution methods that exploit the sparse nature of finite element equations. Such methods include the frontal solution and Cholesky (skyline) factorization for direct solutions, conjugate gradient methods for iterative solutions and the Lanczos method and subspace iteration (simultaneous vector iteration) for eigenvalue solutions.

SPECTRAL DENSITY

The Fourier transform of the correlation function. In random vibrations it gives a measure of the significant frequency content in a system. White noise has a constant spectral density for all frequencies.

SPLINE CURVES

A curve fitting technique that preserves zero, first and second derivative continuity across segment boundaries.

SPURIOUS CRACKS

Cracks that appear in a mesh when the elements are not correctly connected together. This is usually an error in the mesh generation process.

STATIC ANALYSIS

Analysis of stresses and displacements in a structure when the applied loads do not vary with time.

STATICALLY DETERMINATE STRUCTURE

A structure where all of the unknowns can be found from equilibrium considerations alone.

STATICALLY EQUIVALENT LOADS

Equivalent nodal loads that have the same equilibrium resultants as the applied loads but do not necessarily do the same work as the applied loads.

STATICALLY INDETERMINATE STRUCTURE REDUNDANT

A structure where all of the unknowns can not be found from equilibrium considerations alone. The compatibility equations must also be used. In this case the structure is said to be redundant.

STATIONARY RANDOM EXCITATION

A force or response that is random but its statistical characteristics do not vary with time.

STEADY-STATE HEAT TRANSFER

Determination of the temperature distribution of a mechanical part having reached thermal equilibrium with the environmental conditions. There are no time varying changes in the resulting temperatures.

STEADY STATE RESPONSE

The response of the system to a periodic forcing function when all of the transient components of the response have become insignificant.

STEP-BY-STEP INTEGRATION

Methods of numerically integrating time varying equations of motion. These methods can be either explicit or implicit.

STIFFNESS

A set of values which represent the rigidity or softness of a particular element. Stiffness is determined by material type and geometry.

STIFFNESS MATRIX

The parameter(s) that relate the displacement(s) to the force(s). For a discrete parameter multi degree of freedom model this is usually given as a stiffness matrix.

STRAIN

A dimensionless quantity calculated as the ratio of deformation to the original size of the body.

STRAIN ENERGY

The energy stored in the system by the stiffness when it is displaced from its equilibrium position.

STRESS

The intensity of internal forces in a body (force per unit area) acting on a plane within the material of the body is called the stress on that plane.

STRESS ANALYSIS

The computation of stresses and displacements due to applied loads. The analysis may be elastic, inelastic, time dependent or dynamic.

STRESS AVERAGING STRESS SMOOTHING

The process of filtering the raw finite element stress results to obtain the most realistic estimates of the true state of stress.

STRESS CONCENTRATION

A local area of the structure where the stresses are significantly higher than the general stress level. A fine mesh of elements is required in such regions if accurate estimates of the stress concentration values are required.

STRESS CONTOUR PLOT

A plot of a stress component by a series of color filled contours representing regions of equal stress.

STRESS DISCONTINUITIES STRESS ERROR ESTIMATES

Lines along which the stresses are discontinuous. If the geometry or loading changes abruptly along a line then the true stress can be discontinuous. In a finite element solution the element assumptions means that the stresses will generally be discontinuous across element boundaries. The degree of discontinuity can then be used to form an estimate of the error in the stress within the finite element calculation.

STRESS EXTRAPOLATION

The process of taking the stress results at the optimum sampling points for an element and extrapolating these to the element node points.

STRESS INTENSITY FACTORS

A measure of the importance of the stress at a sharp crack tip (where the actual stress values will be infinite) used to estimate if the crack will propagate.

STRESS VECTOR STRESS TENSOR STRAIN VECTOR STRAIN TENSOR

The stress (strain) vector is the components of stress (strain) written as a column vector. For a general three dimensional body this is a (6x1) matrix. The components of stress (strain) written in tensor form. For a general three dimensional body this forms a (3x3) matrix with the direct terms down the diagonal and the shear terms as the off-diagonals.

STRESS-STRAIN LAW

The material property behavior relating stress to strain. For a linear behavior this is Hookes law (linear elasticity). For elastic plastic behavior it is a combination of Hookes law and the Prandtl-Reuss equations.

SUBSPACE VECTOR ITERATION

A method for finding the first few eigenvalues and eigenvectors of a finite element system. This is also known as simultaneous vector iteration.

SUBSTRUCTURING

An efficient way of solving large finite element analysis problems by breaking the model into several parts or substructures, analyzing each one individually, and then combining them for the final results.

SUBSTRUCTURING SUPER ELEMENT METHOD

Substructuring is a form of equation solution method where the structure is split into a series of smaller structures - the substructures. These are solved to eliminate the internal freedoms and the complete problem solved by only assembling the freedoms on the common boundaries between the substructures. The intermediate solution where the internal freedoms of a substructure have been eliminated gives the super element matrix for the substructure.

SURFACE MODELING

The geometric modeling technique in which the model is created in terms of its surfaces only without any volume definition.

TEMPERATURE CONTOUR PLOTS

A plot showing contour lines connecting points of equal temperature.

TETRAHEDRON TETRAHEDRAL ELEMENT

A three dimensional four sided solid element with triangular faces.

THERMAL CAPACITY

The material property defining the thermal inertia of a material. It relates the rate of change of temperature with time to heat flux.

THERMAL CONDUCTIVITY

The material property relating temperature gradient to heat flux.

THERMAL LOADS

The equivalent loads on a structure arising from thermal strains. These in turn arise from a temperature change.

THERMAL STRAINS

The components of strain arising from a change in temperature.

THERMAL STRESS ANALYSIS

The computation of stresses and displacements due to change in temperature.

THIN SHELL ELEMENT THICK SHELL ELEMENT

In a shell element the geometry is very much thinner in one direction than the other two. It can then be assumed stresses can only vary linearly at most in the thickness direction. If the through thickness shear strains can be taken as zero then a thin shell model is formed. This uses the Kirchoff shell theory If the transverse shear strains are not ignored then a thick shell model is formed. This uses the Mindlin shell theory. For the finite element method the thick shell theory generates the most reliable form of shell elements. There are two forms of such elements, the Mindlin shell and the Semi-Loof shell.

TIME DOMAIN

The structures forcing function and the consequent response is defined in terms of time histories. The Fourier transform of the time domain gives the corresponding quantity in the frequency domain.

TRACE OF THE MATRIX

The sum of the leading diagonal terms of the matrix.

TRANSFINITE MAPPING

A systematic method for generating element shape functions for irregular node distributions on an element.

TRANSFORMATION METHOD

Solution techniques that transform coordinate and force systems to generate a simpler form of solution. The eigenvectors can be used to transform coupled dynamic equations to a series of single degree of freedom equations.

TRANSIENT FORCE

A forcing function that varies for a short period of time and then settles to a constant value.

TRANSIENT RESPONSE

The component of the system response that does not repeat itself regularly with time.

TRANSITION ELEMENT

Special elements that have sides with different numbers of nodes. They are used to couple elements with different orders of interpolation, typically a transition element with two nodes on one edge and three on another is used to couple a 4-node quad to an 8-node quad.

TRANSIENT HEAT TRANSFER

Heat transfer problems in which temperature distribution varies as a function of time.

TRIANGULAR ELEMENTS

Two dimensional or surface elements that have three edges.

TRUSS ELEMENT

A one dimensional line element defined by two nodes resisting only axial loads.

ULTIMATE STRESS

The failure stress (or equivalent stress) for the material.

UNDAMPED NATURAL FREQUENCY

The square root of the ratio of the stiffness to the mass (the square root of the eigenvalue). It is the frequency at which an undamped system vibrates naturally. A system with n degrees of freedom has n natural frequencies.

UNDER DAMPED SYSTEM

A system which has an equation of motion where the damping is less than critical. It has an oscillatory impulse response.

UNIT MATRIX

A diagonal matrix with unit values down the diagonal.

UPDATED LAGRANGIAN TOTAL LAGRANGIAN

The updated Lagrangian coordinate system is one where the stress directions are referred to the last known equilibrium state. The total Lagrangian coordinate system is one where the stress directions are referred to the initial geometry.

UPWINDING IN FLUIDS

A special form of weighting function used in viscous flow problems (solution to the Navier-Stokes equations) used in the weighted residual method to bias the results in the direction of the flow.

VARIABLE BANDWIDTH (SKYLINE)

A sparse matrix where the bandwidth is not constant. Some times called a skyline matrix.

VELOCITY

The first time derivative of the displacement.

VIRTUAL CRACK EXTENSION CRACK PROPAGATION

A technique for calculating the energy that would be released if a crack increased in size. This gives the energy release rate which can be compared to the critical energy release (a material property) to decide if a crack will propagate.

VIRTUAL DISPLACEMENTS

An arbitrary imaginary change of the system configuration consistent with its constraints.

VIRTUAL WORK VIRTUAL DISPLACEMENTS VIRTUAL FORCES

Techniques for using work arguments to establish equilibrium equations from compatibility equations (virtual displacements) and to establish compatibility equations from equilibrium (virtual forces).

VISCOUS DAMPING

The damping is viscous when the damping force is proportional to the velocity.

VISCOUS DAMPING MATRIX

The matrix relating a set of velocities to their corresponding velocities

VOLUME DISTORTION VOLUMETRIC DISTORTION

The distortion measured by the determinant of the Jacobian matrix, det j.

VON MISES STRESS

An "averaged" stress value calculated by adding the squares of the 3 component stresses (X, Y and Z directions) and taking the square root of their sums. This value allows for a quick method to locate probable problem areas with one plot.

VON MISES EQUIVALENT STRESS TRESCA EQUIVALENT STRESS

Equivalent stress measures to represent the maximum shear stress in a material. These are used to characterize flow failures (e.g. plasticity and creep). From test results the Von-Mises form seems more accurate but the Tresca form is easier to handle.

WAVE PROPAGATION

The dynamic calculation involving the prediction of the history of stress and pressure waves in solids and fluids.

WAVEFRONT (FRONT)

The wavefront of a symmetric matrix is the maximum number of active nodes at any time during a frontal solution process. It is a measure of the time required to factorise the equations in a frontal solution. It is minimized be element renumbering.

WEIGHTED RESIDUALS

A technique for transforming a set of partial differential equations to a set of simultaneous equations so that the solution to the simultaneous equations satisfy the partial differential equations in a mean sense. The form used in the finite element method is the Galerkin process. This leads to identical equations to those from virtual work arguments.

WHIRLING STABILITY

The stability of rotating systems where centrifugal and Coriolis are also present.

WHITE NOISE

White noise has a constant spectral density for all frequencies.

WILSON THETA METHOD

An implicit solution method for integrating second order equations of motion. It can be made unconditionally stable.

WORD LENGTH

Within a digital computer a number is only held to a finite number of significant figures. A 32bit (single precision) word has about 7 significant figures. A 64bit (double precision) word has about 13 significant figures. All finite element calculations should be conducted in double precision.

YOUNGS MODULUS

The material property relating a uniaxial stress to the corresponding strain.

ZERO ENERGY MODES ZERO STIFFNESS MODES

Non-zero patterns of displacements that have no energy associated with them. No forces are required to generate such modes, Rigid body motions are zero energy modes. Buckling modes at their buckling loads are zero energy modes. If the elements are not fully integrated they will have zero energy displacement modes. If a structure has one or more zero energy modes then the matrix is singular.