# A review paper on Finite elements methods using ANSYS

## B.V.H.S. SAI[1]

Department of Mechanical Engineering, Mahatma Gandhi Institute of Technology, Hyderabad, India[1]

**Abstract:** The aim of this review paper is to represent a general study on the finite element methods using ANSYS software. The various options involved in the ANSYS software are studied in detail. In general form the finite element method is a computational procedure used to resolve tensor states in three dimensional continua based upon the conditions at the boundary. The procedure provides approximate solutions for continua of any shape or configuration, some of which may be very difficult or impossible to solve in closed form. This facility is acquired by the fact that the procedure divides a continuum into smaller units, often many units, and the shape of the units are selected to be shapes for which solutions are either known or can be solved by approximate numerical methods. The elastic equations for each of the small units are assembled for simultaneous solution by matrix methods, thereby solving for the boundary conditions (displacements) of each small unit that are consistent with the conditions at the boundary of the continuum.

## INTRODUCTION

ANSYS is an engineering simulation software (computer-aided engineering, or CAE) developer that is headquartered south of Pittsburgh in Canonsburg, Pennsylvania, United States.

ANSYS was listed on the NASDAQ stock exchange in 1996. In late 2011, Investor's Business Daily ranked ANSYS as one of only six technology businesses worldwide to receive the highest possible score on its Smart Select Composite Ratings. ANSYS has been recognized as a strong performer by a number of other sources as well. The organization reinvests 15 percent of its revenues each year into research to continually refine the software.

**The ANSYS Workbench Interface**

The ANSYS Workbench interface consists primarily of a Toolbox region, the Project Schematic, the Toolbar, and the Menu bar. Depending on the analysis type and/or application or workspace, you may also see other windows, tables, charts, etc. One way to work in ANSYS Workbench is to drag an item such as a component or analysis system from the Toolbox to the Project Schematic or to double-click on an item to initiate the default action. You can also use the context menus, accessible from a right-mouse click, for additional options. You will view your analysis systems -- the components that make up your analysis -- in the Project Schematic, including all connections and links between the systems. The individual applications in which you work will display separately from the ANSYS Workbench GUI, but the results of the actions you take in the applications may be reflected in the Project Schematic.
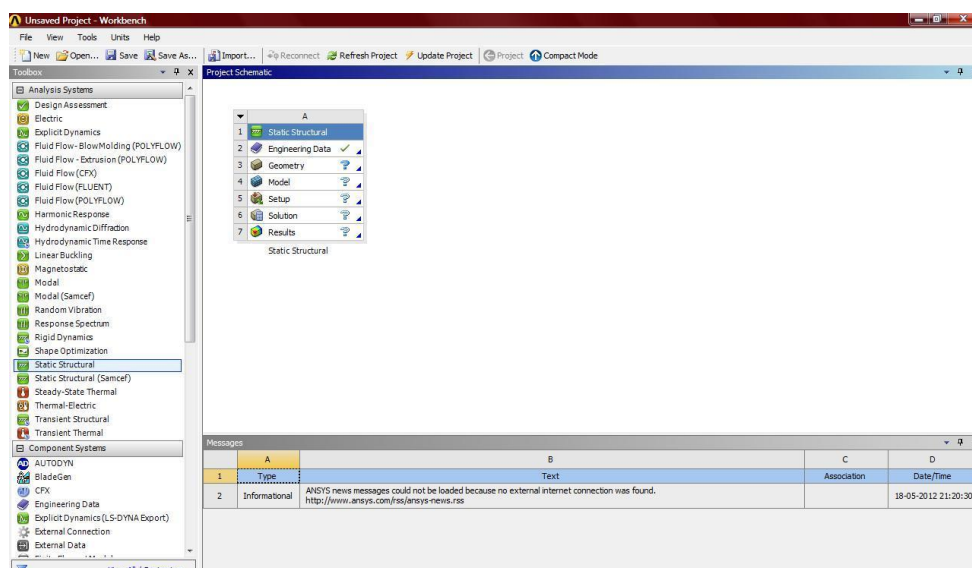
The following GUI components are discussed in more detail:

- Toolbox
- Project Schematic
- The Toolbar
- The Menu Bar
- Working with Views and Workspaces
- Using the Chart View
- Working with Units

**Toolbox**

The ANSYS Workbench Toolbox presents the types of data that you can add to your project. The Toolbox is context-sensitive; as you select different items in the Project Schematic or other workspaces, the contents of the Toolbox may change to reflect the components and actions available to you. When working in other workspaces, such as Engineering Data or Parameters, you can return to the Project Workspace by clicking the 'Return to Project' button on the Toolbar. When you initially enter ANSYS Workbench, the Toolbox contains the system templates that you can add to your project (including standard and custom templates). To build a project, drag templates onto the Project Schematic or double-click to add a new template. Dragging a template onto the Project Schematic allows you to choose where to place the template, and based on your choice, automatically creates the appropriate data shares and transfers. Double-clicking will always add a new system (or systems) and will never create data shares or transfers with any existing systems. See Working Through an Analysis System for more detailed information on building and linking systems.

By default, systems are arranged by system type into one of the following groups:

- Analysis Systems -- You will typically use this core set of templates to build your project. Place these templates in the project schematic to create a system or linked systems with all the necessary components (geometry through post-processing) defined and ready to be populated.
- Component Systems -- These system templates are generally used as component building blocks of a project and represent only a subset of a complete analysis. For example, use a Geometry system to define your geometry. That system can then be connected to several downstream systems, indicating that the downstream systems share the same geometry source. These system templates also include data-integrated software programs that exist outside ANSYS Workbench as independent applications, allowing you to use ANSYS Workbench to manage your analysis data and files. This feature is useful for products such as the Mechanical APDL application, which uses numerous files during an analysis.
- Custom Systems -- Initially, this group contains a selected set of pre-defined coupled system templates. These systems contain more than one system block with predefined data connections. You can also create your own project templates, which will then be stored here.
- Design Exploration -- Use these options to perform various Design Exploration studies. When added to the Project Schematic, each of these systems will be connected by a Parameter Set bus bar.

## PROJECT SCHEMATIC

The Project Schematic captures the structure and workflow of your project, providing a visual representation of the objects in the project and their relationship to each other. Analyses are shown as systems, which are comprised of individual cells.

Projects can vary in complexity, from a single system representing all the necessary steps for a desired analysis, to a complex set of connected (linked) systems representing coupled analysis or variations in modeling approaches.

For example, the project schematic shown below represents a coupled fluid/structural analysis, and the links between the systems represent data relationships. The link with the square terminator indicates that the two systems share a common geometry. The link with the round terminator indicates that solution data from the Fluid Flow system is transferred as a

load to the setup of the static structural system.

To start a project, drag a template or object from the Toolbox into the Project Schematic or right-click in the Schematic white space and select an analysis type from the context menu. As you build your project, associations and connections will be automatically maintained and updated.

See Working Through an Analysis System for more detailed information on building and linking systems.

The following Project Schematic topics are discussed in more detail:

- Systems and Cells
- Project Schematic Links
- Context Menu Options

## SYSTEMS AND CELLS

Each item that you add to your project from the Toolbox is represented as a system. A system is comprised of individual components called cells. To define the details of a simulation, you generally interact with systems at the cell level. Use a right mouse click to expose a context-sensitive menu of options, called a context menu, specific to that system or cell. Double-click a cell to initiate the default action (shown in the context menu in bold). Via a cell, you might:

- launch a data-integrated application or workspace
- add connecting systems, either upstream or downstream
- assign input or reference files
- assign properties to components of your analysis

Each cell has an application or workspace associated with it. Some cells are associated with data-integrated applications such as, ANSYS FLUENT or the Mechanical application; these applications launch in a separate window. Some cells, such as Engineering Data or Parameters, are associated with workspaces (an arrangement of views contained within the Workbench window, see Working with Views and Workspaces ). In some cases, multiple cells in a system will be associated with the same application.

Icons for each cell indicate that particular cell's state (i.e., needs attention, up to date, etc.). See Understanding Cell States for a detailed explanation of states.

The example Modal system shown illustrates cells in different states.

To display a quick help panel for the cell, left- or right-mouse click on the blue triangle in the lower right corner of the cell (where available). The quick help will explain any immediate action that needs to be taken and includes links to more detailed help.

The following Systems and Cells topics are discussed below:

- Types of Cells
- Understanding Cell States

**Types of Cells**

The following common types of cells occur in many of the analysis and component systems available in ANSYS Workbench; how you work with them is explained below. Other cell types may be available in certain systems; see the application-specific documentation under Systems for these cell descriptions.

- Engineering Data
- Geometry
- Model/Mesh
- Setup

- Solution
- Results

**Engineering Data**

Use the Engineering Data cell with Mechanical systems or the Engineering Data component system to define or access material models for use in an analysis. Double-click the Engineering Data cell, or right-mouse click and choose **Edit** from the context menu to display the Engineering Data workspace to define material data. For more information, see Engineering Data.

**Geometry**

Use the Geometry cell to import, create, edit or update the geometry model used for analysis. Right-mouse click to display the context menu to access these functions. The right mouse button options are context sensitive and will change as the state of your geometry changes. All geometry-specific options are described here; not all will be always available. These options are in addition to the common options described in Common Context Menu Options and Transfer Context Menu Options. The Space Claim Direct Modeler features are available only if you have Space Claim Direct Modeler installed and the ANSYS Space Claim Direct Modeler license available.

New Geometry (or New Design Modeler Geometry/New Space Claim Direct Modeler Geometry)

Launches Design Modeler or Space Claim Direct Modeler, where you can build a new geometry.

Import Geometry

Select Browse to open a dialog box that allows you to navigate to an existing geometry file, or select a file from the list of recently viewed files.

Edit

After you have attached a geometry to your system by choosing either **New Geometry** or **Import Geometry**, click **Edit** to open the model in Design Modeler or Space Claim Direct Modeler to modify it.

Replace Geometry

Select Browse to open a dialog box that allows you to navigate to an existing geometry file or select a file from the list of recently viewed files to replace the currently specified file.

Update from CAD

Generates an existing CAD geometry using the parameter values as defined in the CAD system.

Refresh

Reads in all modified upstream data but does not regenerate the geometry. Enabled when the Geometry cell is in the Refresh Required state.

Properties

Displays a Properties pane where you can select basic and advanced geometry properties. For a detailed description of the options available from the Properties pane, see Geometry Preferences in the CAD Integration section of the ANSYS Workbench help.

## MODEL/MESH

The Model cell in the Mechanical application analysis systems or the Mechanical Model component system is associated with the Model branch in the Mechanical application and affects the definition of the geometry, coordinate systems, connections and mesh branches of the model definition. When linking two systems, you cannot create a share between the Model cells of two established systems. You can generate a second system that is linked at the Model cell of the first system, but you cannot add a share after the second system has been created. Likewise, you cannot delete a link between the Model cells of two systems.

The Mesh cell in Fluid Flow analysis systems or the Mesh component system is used to create a mesh using the Meshing application. It can also be used to import an existing mesh file.

Edit

Launches the appropriate Model or Mesh application (the Mechanical application, Meshing, etc.)

**Setup**

Use the Setup cell to launch the appropriate application for that system. You will define your loads, boundary conditions, and otherwise configure your analysis in the application. The data from the application will then be incorporated in the project in ANSYS Workbench, including connections between systems.

For the Mechanical application systems, you will see the following Setup options, in addition to the common options:

Edit

Launches the Mechanical application with the geometry loaded and with cells mapped to their respective tree locations in the Mechanical application.

For CFX systems, you will see the following Setup options, in addition to the common options:

Edit

Launches CFX-Pre.

Import Case

Imports an existing case file containing physics data, region and mesh information for your analysis.

For FLUENT systems, you will see the following Setup options, in addition to the common options:

Edit

Launches ANSYS FLUENT.

Import Case

Imports an existing FLUENT case file.

**Solution**

From the Solution cell, you can access the Solution branch of your application, and you can share solution data with other downstream systems (for instance, you can specify the solution from one analysis as input conditions to another analysis). If you have an analysis running as a remote process, you will see the Solution cell in a pending state until the remote process completes. See the discussion on Understanding Cell States, below.

For the Mechanical application systems, you will see the following Setup options, in addition to the common options described earlier:

Edit

Launches the Mechanical application open to the Solution branch.

Delete

Deletes the Solution and Results cell. Deleting the solution cell makes the system a setup-only system, meaning the system will generate only an input file. It will not solve or post results. The Solution object and below are removed from the Mechanical application tree.

For CFX systems, you will see the following Solution options, in addition to the common options:

Edit

Launches CFX-Solver Manager.

Import Solution

Displays the **Open** dialog, where you can specify the CFX Solver Results file to load. When the results file is loaded, the system will display only the **Solution** cell and the **Results** cell.

Display Monitors

Opens the ANSYS CFX-Solver Manager and shows the results of the previous run.

For FLUENT systems, you will see the following Solution options, in addition to the common options:

Edit

Launches ANSYS FLUENT.

Import Initial Data

Allows you to select an existing FLUENT data file to use for initialization.

**2.5.6 Results**

The Results cell indicates the availability and status of the analysis results (commonly referred to as post-processing). From the Results cell, you cannot share data with any other system.

## FINITE ELEMENT METHOD

A finite element method (abbreviated as FEM) is a numerical technique to obtain an approximate solution to a class of problems governed by elliptic partial differential equations. Such problems are called as boundary value problems as they consist of a partial differential equation and the boundary conditions. The finite element method converts the elliptic partial differential equation into a set of algebraic equations which are easy to solve. The initial value problems which consist of a parabolic or hyperbolic differential equation and the initial conditions (besides the boundary conditions) cannot be completely solved by the finite element method. The parabolic or hyperbolic differential equations contain the time as one of the independent variables. To convert the time or temporal derivatives into algebraic expressions, another numerical technique like the finite difference method (FDM) is required. Thus, to solve an initial value problem, one needs both the finite element method as well as the finite difference method where the spatial derivatives are converted into algebraic expressions by FEM and the temporal derivatives are converted into algebraic equations by FDM.

There are two types of integral formulations :

- Weak or Weighted residual formulation
- Variational formulation

In finite element method, the solution of a boundary value problem is obtained by using one of these two integral formulations. When it is difficult to solve the differential equation of a boundary value problem, this method provides an alternative way to obtain the solution. But, usually, it is an approximate solution.

In Ritz method, an approximate solution of a boundary value problem is obtained by using the corresponding integral formulation. An approximate form for the solution is assumed in terms of a series containing known functions and unknown coefficients. When this form is substituted in the integral formulation, we get a set of algebraic equations in terms of the unknown coefficients. Solution of the algebraic equations determines the coefficients. To get the exact solution, usually, an infinite series is needed. A finite series, normally, gives an approximate solution. However, when sufficiently large number of terms is chosen, the accuracy is reasonable for most engineering applications.

### How does the Finite Element Method Work?

General techniques and terminology of finite element analysis will be introduced with reference to Figure. The figure depicts a volume of some material or materials having known physical properties. The volume represents the domain of a boundary value problem to be solved. For simplicity, at this point, we assume a two-dimensional case with a single field variable f(x, y) to be determined at every point P(x, y) such that a known governing equation (or equations) is satisfied exactly at every such point. Note that this implies an exact
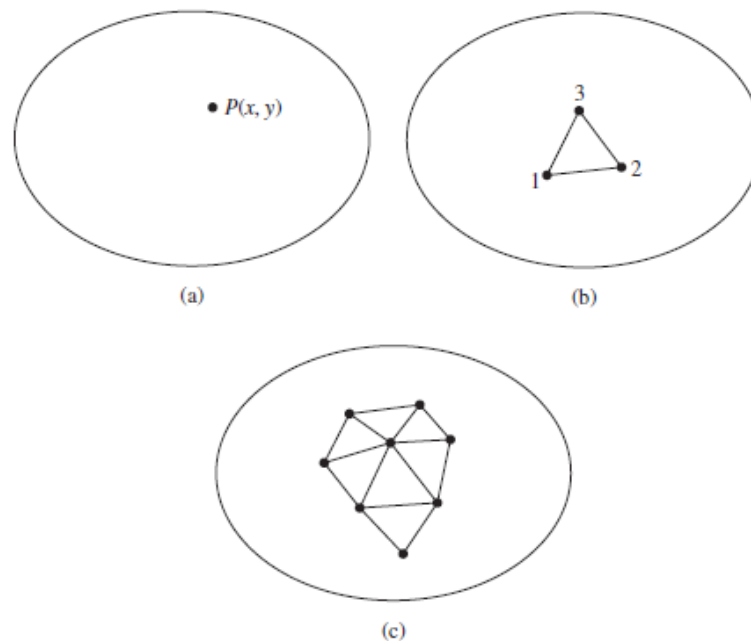


### Finite Element Method

The solution is a closed-form algebraic expression of the independent variables. In practical problems, the domain may be geometrically complex as is, often, the governing equation and the likelihood of obtaining an exact closed-form solution is very low. Therefore, approximate solutions based on numerical techniques and digital computations are most often obtained in engineering analyses of complex problems. Finite element analysis is a powerful technique for obtaining such approximate solutions with good accuracy.

A small triangular element that encloses a finite-sized sub domain of the area of interest is shown in Figure b. That this element is not a differential element of size dx × dy makes this a finite element. As we treat this example as a two-dimensional problem, it is assumed that the thickness in the z direction is constant and z dependency is not indicated in

the differential equation. The vertices of the triangular element are numbered to indicate that these points are nodes. A node is a specific point in the finite element at which the value of the field variable is to be explicitly calculated. Exterior nodes are located on the boundaries of the finite element and may be used to connect an element to adjacent finite elements. Nodes that do not lie on element boundaries are interior nodes and cannot be connected to any other element. The triangular element of Figure has only exterior nodes.

If the values of the field variable are computed only at nodes, how are values obtained at other points within a finite element? The answer contains the crux of the finite element method: The values of the field variable computed at the nodes are used to approximate the values at no nodal points (that is, in the element interior) by interpolation of the nodal values. For the three-node triangle example, the nodes are all exterior and, at any other point within the element, the field variable is described by the approximate relation

$$f(x, y) = N_1 f(x, y)_1 + N_2 f(x, y)_2 + N_3 f(x, y)_3$$

where $f_1$, $f_2$, and $f_3$ are the values of the field variable at the nodes, and $N_1$, $N_2$, and $N_3$ are the interpolation functions, also known as shape functions or blending functions. In the finite element approach, the nodal values of the field variable are treated as unknown constants that are to be determined. The interpolation functions are most often polynomial forms of the independent variables, derived to satisfy certain required conditions at the nodes. These conditions are discussed in detail in subsequent chapters. The major point to be made here is that the interpolation functions are predetermined, known functions of the independent variables; and these functions describe the variation of the field variable within the finite element. The triangular element described by Equation is said to have 3 degrees of freedom, as three nodal values of the field variable are required to describe the field variable everywhere in the element. This would be the case if the field variable represents a scalar field, such as temperature in a heat transfer problem . If the domain of Figure represents a thin, solid body subjected to plane stress (Chapter 9), the field variable becomes the displacement vector and the values of two components must be computed at each node. In the latter case, the three-node triangular element has 6 degrees of freedom. In general, the number of degrees of freedom associated with a finite element is equal to the product of the number of nodes and the number of values of the field variable (and possibly its derivatives) that must be computed at each node.

How does this element-based approach work over the entire domain of interest?

As depicted in Figure c, every element is connected at its exterior nodes to other elements. The finite element equations are formulated such that, at the nodal connections, the value of the field variable at any connection is the same for each element connected to the node. Thus, continuity of the field variable at the nodes is ensured. In fact, finite element formulations are such that continuity of the field variable across inter element boundaries is also ensured. This feature avoids the physically unacceptable possibility of gaps or voids occurring in the domain. In structural problems, such gaps would represent physical separation of the material. In heat transfer, a "gap" would manifest itself in the form of different temperatures at the same physical point.

Although continuity of the field variable from element to element is inherent to the finite element formulation, inter element continuity of gradients (i.e., derivatives) of the field variable does not generally exist. This is a critical observation. In most cases, such derivatives are of more interest than are field variable values. For example, in structural problems, the field variable is displacement but the true interest is more often in strain and stress. As strain is defined in terms of first derivatives of displacement components, strain is not continuous across element boundaries. However, the magnitudes of discontinuities of derivatives can be used to assess solution accuracy and convergence as the number of elements is increased.

**General procedure for Finite Element Method**

Certain steps in formulating a finite element analysis of a physical problem are common to all such analyses, whether structural, heat transfer, fluid flow, or some other problem. These steps are embodied in commercial finite element software packages (some are mentioned in the following paragraphs) and are implicitly incorporated in this text, although we do not necessarily refer to the steps explicitly in the following chapters.

The steps are described as follows.

**Pre-processing**

The pre-processing step is, quite generally, described as defining the model and includes

- Define the geometric domain of the problem.
- Define the element type(s) to be used.
- Define the material properties of the elements.
- Define the geometric properties of the elements (length, area, and the like).
- Define the element connectivity's (mesh the model).
- Define the physical constraints (boundary conditions).

- Define the loadings.

The pre-processing (model definition) step is critical. In no case is there a better example of the computer-related axiom "garbage in, garbage out." A perfectly computed finite element solution is of absolutely no value if it corresponds to the wrong problem.

## Solution

During the solution phase, finite element software assembles the governing algebraic equations in matrix form and computes the unknown values of the primary field variable(s). The computed values are then used by back substitution to compute additional, derived variables, such as reaction forces, element stresses, and heat flow. As it is not uncommon for a finite element model to be represented by tens of thousands of equations, special solution techniques are used to reduce data storage requirements and computation time. For static, linear problems, a wave front solver, based on Gauss elimination is commonly used.

## Post processing

Analysis and evaluation of the solution results is referred to as post processing. Postprocessor software contains sophisticated routines used for sorting, printing, and plotting selected results from a finite element solution. Examples of operations that can be accomplished include.

- Sort element stresses in order of magnitude.
- Check equilibrium.
- Calculate factors of safety.
- Plot deformed structural shape.
- Animate dynamic model behaviour.
- Produce colour-coded temperature plots.

While solution data can be manipulated many ways in post processing, the most important objective is to apply sound engineering judgment in determining whether the solution results are physically reasonable.

## REFERENCES

[1]. Bose, Kunal K; Penchaliah, Ramkumar, (2019). Finite element method based sliding wear prediction of steel-on-steel contacts using extrapolation techniques. Proceedings of the Institution of Mechanical Engineers, Part J: Journal of Engineering Tribology, pp. 1-18.

[2]. S. Nemat-Nasser (1979). Decomposition of strain measures and their rates in finite deformation elastoplasticity. 15(2), pp.155–166.

[3]. Griffin, Jerry H. (1990). A Review of Friction Damping of Turbine Blade Vibration. International Journal of Turbo and Jet Engines, 7(3-4), pp. 297-307.

[4]. Ender Cigeroglu; Wangming Lu; Chia-Hsiang Menq (2006). One-dimensional dynamic microslip friction model. 292(3-5), pp. 881–898.

[5]. Marquina, Francisco J.; Coro, Armando; Gutiérrez, Alberto; Alonso, Roberto; Ewins, David J.; Girini, Giovanna (2008). [ASME ASME Turbo Expo 2008: Power for Land, Sea, and Air - Berlin, Germany (June 9–13, 2008)] Volume 5: Structures and Dynamics, Parts A and B - Friction Damping Modeling in High Stress Contact Areas Using Microslip Friction Model. pp. 309–318.

[6] Demirhan, Necdet; Kanber, Bahattin (2013). Finite Element Analysis of Frictional Contacts of FGM Coated Elastic Members, Mechanics Based Design of Structures and Machines, 41(4), pp. 383–398.