Finite Element Method Crash Course with ANSYS

P. Dechaumphai
S. Sucharitpwatskul

Finite Element Method
Crash Course with ANSYS

Finite Element Method
Crash Course with ANSYS

Pramote Dechaumphai

Sedthawatt Sucharitpwatskul

2026

179.-

Finite Element Method Crash Course with ANSYS
e-Book Edition
280 pages

Printed from the camera-ready copy provided by the author.

Published by
Dr. Pramote Dechaumphai
International School of Engineering (ISE)
Faculty of Engineering, Chulalongkorn University
Phyathai Road, Pathumwan, Bangkok 10330
Tel: (662) 218-6621

Distributed by
Chulalongkorn University Book Center
Phyathai Road, Pathumwan, Bangkok 10330
Tel: (662) 255-4433
Website: chulabook.com

Proofreader: Professor Dr. Pramote Dechaumphai
Book Layout Design and Typesetting: Kaiumporn Phongkhachorn

Preface

The book "*Finite Element Method Crash Course with Ansys*" is written for users to learn the finite element method and to use the latest Ansys software version quickly. The book is also suitable for designers and engineers before using the software to analyze realistic problems.

The book contains twelve chapters describing different analyses of engineering problems. These problems are in the fields of solid mechanics, heat transfer and fluid flows. In each chapter, the governing differential equations and the finite element method are presented. An academic example is used to demonstrate the Ansys procedure for solving it in detail. An application example is also included at the end of each chapter to highlight the software capability for analyzing realistic problems.

The authors would like to thank the ANSYS, Inc., USA for providing the software to prepare this book. The authors appreciate Dr. Edward Warute Dechaumphai for proof-reading the book manuscript.

Pramote Dechaumphai
Sedthawatt Sucharitpwatskul

Contents

Chapter 1

Introduction

1.1 Solving Engineering Problems

Computer-Aided Engineering (CAE) has played an important role in engineering design and analysis. Designers and engineers nowadays use CAE software packages to improve their product quality. The software packages help reducing designed time and material consumption while increasing the product strength and life time. Trial-and-error process, based solely on intuition of designers and engineers, is minimized or eliminated.

Most of CAE software packages employ the finite element and finite volume methods to provide design and analysis solutions. These methods are based on engineering mathematics together with the application of numerical methods. The output numerical solutions are converted and displayed graphically so that the simulated results can be understood easily. Without knowing how the software solves the problem, it is difficult for new users to be confident with the validity of output solutions.

Mathematics and engineering governing equations embedded in these CAE software packages represent the nature of the problem being considered. As an example of fluid flow problem, mass and momentums must be conserved at any location in the flow domain. Such conservations are expressed in form of partial differential equations that are taught in fluid flow courses. This means users should have some background in mathematics together with the understanding of their physical meanings. By employing the finite volume method, these partial differential equations are transformed into a large set of algebraic equations. A computer program is developed to solve these algebraic equations for the flow solutions. The computed solutions are displayed as color graphics on computer screen.

Similarly, users need to understand the equilibrium equations before analyzing a structural problem. These equilibrium equations are again in form of the partial differential equations as seen in many solid mechanics textbooks. The finite element method transforms these differential equations into their corresponding algebraic equations. A computer program is developed to solve such algebraic equations for the deformed shape and stresses that occur in the structure.

The explanation above indicates that users should have backgrounds in mathematics and physics of the problem being solved. Users are also needed to understand the finite element/ volume method prior to use any CAE software package. They can then convince themselves on the solutions generated by the software. This is one of the main reasons that most universities are offering the finite element/volume method courses to engineering students.

1.1.1 Problem Ingredients

Solutions to an engineering problem depend on the three components:

(a) *Differential Equations*. The differential equations interpret and model physical behavior of the problem into mathematical functions. For example, if we would like to determine temperature distribution of a ceramic cup containing hot coffee, we need to solve the differential equation that describes the conservation of energy at any location on the cup. The differential equation contains partial derivative terms representing conduction heat transfer inside the cup material. Such differential equation is not easy to solve using analytical approaches.

(b) *Boundary Conditions*. The temperature distribution on the cup depends on the coffee temperature inside the cup and the surrounding ambient temperature outside the cup surface. Different boundary conditions thus affect the cup temperature solution.

(c) *Geometry*. Cup shapes also affect their temperature distribution, even though they are made from the same material and placed under the same boundary conditions. The cup temperature changes if the cup is larger or thicker.

The three components above always affect the solutions of the problem being solved. In undergraduate classes, we learned how to solve simplified forms of differential equations subjected to simple boundary conditions on plain geometries to obtain exact or analytical solutions. For real-life practical problems, they are governed by coupled differential equations which are quite sophisticated. Their boundary conditions and geometries are complicated. Numerical methods such as the finite element and finite volume methods are employed to provide accurate approximated solutions.

1.1.2 Solution Methods

Methods for finding solutions can be categorized into two types:

(a) *Analytical Method*. The analytical method herein refers to a mathematical technique used to find an exact or analytical solution for a given problem. The technique can provide solutions only for simple problems as taught in undergraduate courses where differential equations, boundary conditions and geometries are not complicated. Most problems are limited in one dimension so that their governing equations can be simplified from partial to ordinary differential equations.

(b) *Numerical Method*. If the differential equations, boundary conditions and geometry of a given problem are complicated, solving with analytical method is not feasible. We need to find an approximate solution from a numerical method. There are many numerical techniques for finding solutions to complex problems. The popular techniques widely used are the finite element and finite volume methods. This is mainly because both techniques can handle problems with complex geometry effectively.

Both the finite element and finite volume methods transform the governing differential equations into algebraic equations. In the process, many numerical techniques are needed. The techniques include solving a large set of algebraic equations, understanding concepts of the interpolation functions, determining derivatives and integrations of functions numerically, etc. Details of these techniques are taught in undergraduate numerical method courses and can be found in many introductory numerical method textbooks.

1.2 Finite Element Method

Because most of CAE commercial software packages employ the finite element method to solve for solutions, we will introduce the method in this section.

1.2.1 What is the Finite Element Method?

The finite element method is a numerical technique for finding approximated solutions of problems in science and engineering. These problems are governed by the three components including differential equations, boundary conditions and geometries.

The method starts by dividing the problem domain or geometry into a number of small elements. These elements are connected via nodes where the unknowns are to be determined. The finite element equations for each element are derived from the governing differential equations describing physics. These finite element equations are assembled into a large set of algebraic equations. The boundary conditions are then imposed to the set of algebraic equations to solve for solutions at each node.

We will understand the procedure of the finite element method in details in the following section.

1.2.2 Finite Element Method Procedure

The finite element method procedure generally consists of 6 steps:

<u>Step 1</u>: The first step is to construct the domain geometry of the given problem. The geometry may consist of straight lines, curves, circles, surfaces or solid shapes in three dimensions. Different software packages have their unique ways to create geometry. Users may have to spend some times to familiarize with the software. A finite element mesh is then generated on the constructed geometry. Depending on the complexity of the geometry, a mesh may consist of various element types such as line, triangular or brick element. These elements are connected at nodes for which the problem unknowns are located.

<u>Step 2</u>: The second step is to select the element types. For examples, a line element may consist of two or three nodes, or a triangular element may have three or six nodes. The number of element nodes affects the interpolation functions used in that element. Selecting an element with more nodes will increase the number of unknowns and thus the computational time. However, the solution accuracy can also increase when a more complicated interpolation function is used.

<u>Step 3</u>: The third step is the most important step of the finite element method. This step is the derivation of the finite element

equations from the governing differential equations. The derived finite element equations are in the form of algebraic equations that can be computed numerically. The transformation process must be carried out correctly so that the derived algebraic equations can yield accurate solutions.

Step 4: The finite element equations from all elements are then assembled to become a large set of algebraic equations. Assembling element equations must be done properly. This is similar to placing jigsaw pieces at appropriate locations to yield the complete picture.

Step 5: The boundary conditions of the problem are then imposed on the set of algebraic equations before solving for the nodal unknowns. The nodal unknowns are the displacements for structural problem and are the temperatures for heat transfer problem.

Step 6: Other quantities of interest can then be solved. For structural problem, stresses in the structure can be determined after the displacements are known. For heat transfer problem, heat fluxes can be computed once the nodal temperatures are obtained.

The six steps above indicate that the method is quite general and suitable for a large class of problems in science and engineering. The three problem ingredients which are the differential equations, boundary conditions and geometry are handled in the third, fifth and first step of the method, respectively.

1.3 Ansys Software

Ansys software was first developed in 1970 by John Swanson who was an engineer at Westinghouse Astronuclear Laboratory. The software was originally for stress analysis of nuclear reactor components. He later founded Swanson Analysis System, which was named as Ansys, Inc. His Ansys software then became an industry leading finite element program for analyzing engineering problems and optimizing products. At the same time, NASTRAN (NAsa STRuctural ANalysis program) was also popular and being used by NASA Engineers. I remembered Dr. Swanson came to NASA Langley Research Center, Hampton, Virginia to promote his software while I was an engineer there. He gave coffee cups with the early yellow/black Ansys logo to NASA engineers working in the CAE department.

Nowadays, Ansys is a software widely used all over the world for analyzing a large class of problems in many fields. This is mainly because the software is easy to learn and use. Various problems can be solved conveniently while solutions are displayed graphically on the computer screen.

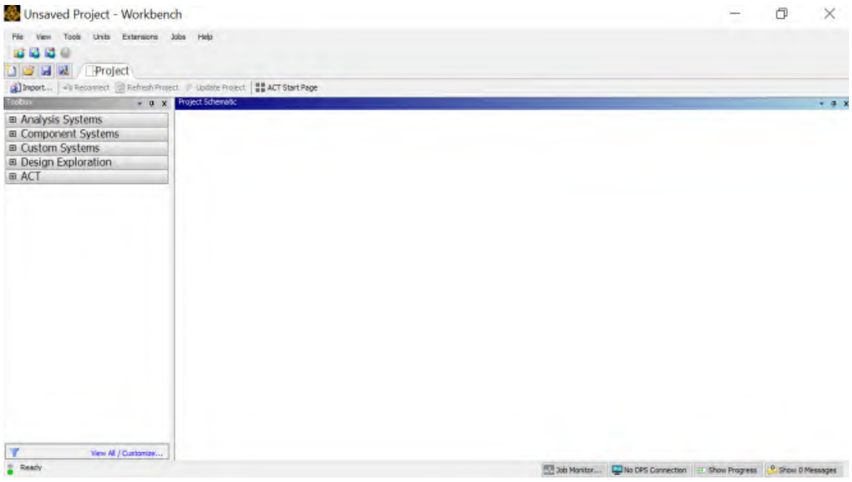
1.3.1 Ansys Workbench

In the early days of Ansys development, the Disk Operating System (DOS) was the most widely used operating system on computers. Ansys users needed to type long and specific commands through keyboards. These commands were required to construct model geometry, such as lines, arcs, surfaces, volumes, etc. Various commands were also needed to create meshes, apply boundary conditions and execute the problem for solutions. Using the software for analyzing a problem at that time was not convenient at all.

Development of Windows environment has provided the ease of using the software. With mouse and keyboard, users can interact with the software graphically. Lately, Ansys has introduced the Workbench function which further simplifies the use of the software via Graphic User Interface (GUI). The Ansys Workbench is employed to solve various types of problems presented as examples in this book.

1.3.2 Screen and Tool Bars

The starting workbench window consists of the menu and tool bars at the top. The large two areas below are the Toolbox and Project Schematic regions as shown in the figure.



The frequently used menu items are:

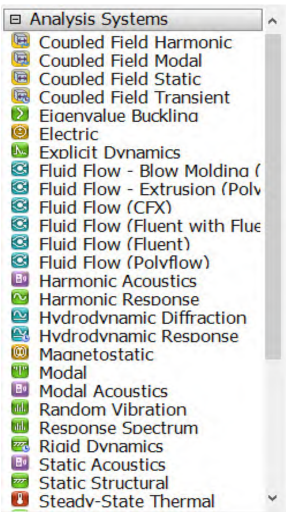
File Create a new file, open an existing file, save the current file, import existing model, etc.

View Arrange the window layout, customize the toolbox, etc.

Tools Set the license preference, select options of appearance, languages, graphics interaction, etc.

Units Select unit systems, define user's units, etc.

Help Get Help from Ansys.

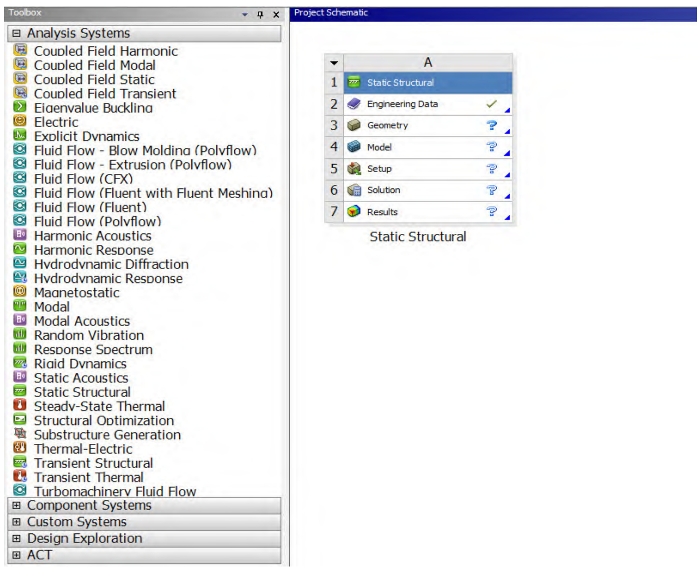


The toolbox region on the left side of the screen contains numerous systems. These include the analysis, component, custom and external connection systems with design exploration. The analysis system consists of several tools for solving different classes of problems such as static and transient structural analyses, buckling and modal analyses, steady-state and transient thermal analyses, fluid flow analysis. These analytical tools are shown in the figure.

The project schematic region on the right side of the screen is the working area. This larger region is for the user to view what is happening at different stages starting from creating geometry domain, discretizing domain into a number of small elements, applying boundary conditions, solving for solutions and displaying results.

1.3.3 Analyzing steps

The analyzing steps via the Workbench follow the standard finite element procedure. As an example of analyzing a static structural problem, we double-click at the **Static Structural** under the **Analysis Systems** in the **Toolbox** window. A small cell of **Static Structural** will appear in the **Project Schematic** window as shown in the figure.



The cell consists of seven items as follows:

1. Static Structural Perform static analysis of a structure.

2. Engineering Data Provide engineering data associated with the problem, such as the material modulus of elasticity, Poisson's ratio, coefficient of thermal expansion, etc.

3. Geometry	Create model geometry of the problem by constructing lines, arcs, circles, surfaces, etc. This step is normally time consuming especially for complex configuration. An imported CAD model file could help reducing the effort.
4. Model	Assign materials and generate a mesh by discretizing the model into a number of small elements. The process is performed automatically.
5. Setup	Specify boundary conditions such as the constraints and loadings, as well as some specific analysis settings.
6. Solution	Solve the problem for solutions. This step is executed automatically if the information provided in the preceding steps is complete.
7. Results	Display solutions in different forms, such as color contours, vectors and surface plots.

The check mark symbol (✓) will appear on the right side of the step if that step has been carried out correctly. Ansys Workbench uses different symbols to explain status of the step as follows:

Nothing is done because upstream data is not available.

Refresh is needed since upstream data has changed.

Attention is required. User interaction is needed.

Update is required because upstream data was modified.

✓ Everything is OK.

Solution is interrupted. Need correction to resume action.

Solution is in progress.

We will follow the above procedure, step by step, to analyze different types of problems in the following chapters. These include structural, heat transfer and fluid flow problems using one-, two- and three-dimensional finite element models. We will find that, if we performed each step correctly, the Ansys Workbench will show the check mark symbol (✔) on the right side of the step. But if we see other symbols, we need to go back and fix that step before moving on. The process thus ensures us that everything has been done appropriately before obtaining the final solutions.

1.4 Advantages of Finite Element Method

The finite element method is popular and widely used by scientists and engineers all over the world for analyzing various types of problems. Examples of problems are as follows.

(a) Stress analyses of large-scale structures such as bridges, ships, trains, aircrafts, automobiles and buildings. Structural analysis for small-scale products are such as automotive and electronic parts, furniture, machine equipment, etc.

(b) Vibration and dynamic analyses of high-voltage power transmission towers, expressway signs under strong wind, crash simulation of automobiles, turbine blades operating under high pressure and temperature, etc.

(c) Fluid analyses of air flows over cities, air ventilation in large halls, inside offices, cleanrooms, computer cases, etc.

(d) Electromagnetic analyses around power transmission lines, electric motors, sensitive electronic devices, etc.

(e) Bio-mechanic analyses of blood flow in human hearts and veins, artificial joints and bones, etc.

(f) Analyses of other problems in which their experiments are dangerous to human or too costly for multiple tests, such as hazardous chemical reaction in gas chambers, prediction of bomb explosion phenomena, flow field around hypersonic aerospace plane, etc.

Advantages of the finite element method as highlighted above have led to many commercial software packages. Users of these software packages must have good background and understanding of the method prior to using them. Basic mathematical theories and the finite element method for structural, heat transfer and fluid flow analyses will be presented in the following chapters with examples. Understanding materials in these chapters is encouraged before using the Ansys Workbench with confidence.

Chapter 2

Truss Analysis

Analysis of truss structures is normally used as the first step toward understanding the finite element method. The analysis is simple because the truss (rod or spring) element contains only a displacement unknown in its axial direction at each node. The finite element equations are easy to derive and problems with few elements can be solved by hands.

2.1 Basic Equations

2.1.1 Differential Equation

A one-dimensional equilibrium equation, in the x-direction of a truss member without the inclusion of its body force, is governed by the equilibrium equation,

$$\frac{\partial \sigma_x}{\partial x} = 0$$

where σ_x is the truss axial stress.

2.1.2 Related Equations

The truss stress varies with the strain ε_x by the Hook's law,

$$\sigma_x = E\,\varepsilon_x$$

where E is the modulus of elasticity or Young's modulus. The strain ε_x is related to the displacement according to the small deformation theory as,

$$\varepsilon_x = \frac{\partial u}{\partial x}$$

where $u = u(x)$ is the displacement that varies with the distance x along the length of the truss member. Thus, the stress can be written in form of the displacement as,

$$\sigma_x = E\frac{\partial u}{\partial x}$$

The governing differential equation, for the case of constant Young's modulus, becomes,

$$E\frac{\partial^2 u}{\partial x^2} = 0$$

For a truss member that lies only in the x-direction, the displacement distribution $u = u(x)$ can be derived from the differential equation above. This is done by performing integrations twice and applying the problem boundary conditions. The stress of the truss member can be then determined. However, if the problem contains many truss members oriented in three dimensions, it is not easy to determine their deformed shape and member stresses. The finite element method offers a convenient way to find the solution as explained in the following section.

2.2 Finite Element Method

2.2.1 Finite Element Equations

Finite element equations can be derived from the governing differential equation by using the Method of Weighted Residuals (MWR). The idea of the method is to transform the differential equation into the corresponding algebraic equations by requiring that the error is minimum. These algebraic equations consist of numerical operations of addition, subtraction, multiplication and division. Such operations allow the use of calculators to determine solutions for small problems. For larger problems, a computer program must be developed and employed.

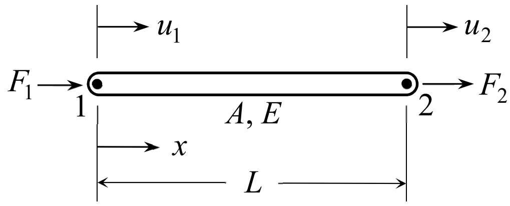
The derived finite element equations are normally written in matrix form so that they can be used in computer programming easily. The finite element equations for the truss element are,

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

where $[K]$ is the element stiffness matrix; $\{u\}$ is the column matrix or vector that consists the nodal displacement unknowns; and $\{F\}$ is the column matrix or vector that contains the nodal loads. These matrices depend on the element types used as explained in the following section.

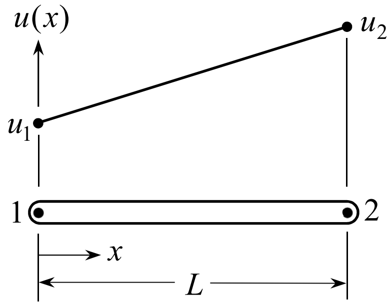
2.2.2 Element Types

The standard two-node truss element is shown in the figure. The element lies in the x-coordinate direction and consists of a node at each end. The element length is L with the cross-sectional area of A and made from a material that has the Young's modulus of E. At an equilibrium condition, the forces at node 1 and 2 are F_1 and F_2, causing the displacements of u_1 and u_2 in its axial direction, respectively.



The displacement distribution is assumed to vary linearly along the element axial x-direction in the form,

$$u(x) = N_1(x)\,u_1 + N_2(x)\,u_2 = \lfloor N_1(x) \quad N_2(x) \rfloor \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix}$$
$$= \underset{(1\times 2)}{\lfloor N(x) \rfloor} \underset{(2\times 1)}{\{u\}}$$

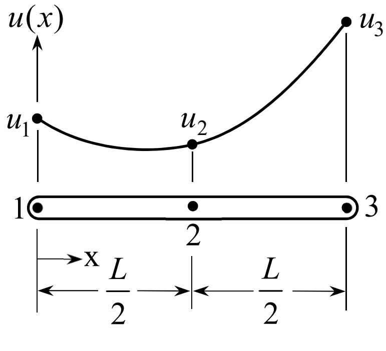


where $N_1(x)$ and $N_2(x)$ are the element interpolation functions. For this two-node element, they are,

$$N_1(x) = 1 - \frac{x}{L}$$

and $$N_2(x) = \frac{x}{L}$$

A truss element may contain more than two nodes. As an example, the three-node truss element, as shown in the figure, assumes the displacement distribution in the form,



$$u(x) = N_1(x)\,u_1 + N_2(x)\,u_2 + N_3(x)\,u_3$$
$$= \lfloor N_1(x) \;\; N_2(x) \;\; N_3(x) \rfloor \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix}$$
$$= \underset{(1\times 3)}{\lfloor N(x) \rfloor} \underset{(3\times 1)}{\{u\}}$$

where $N_1(x)$, $N_2(x)$ and $N_3(x)$ are the interpolation functions expressed by,

$$N_1(x) = 1 - \frac{3x}{L} + \frac{2x^2}{L^2} \quad ; \quad N_2(x) = \frac{4x}{L} - \frac{4x^2}{L^2} \quad ;$$

$$N_3(x) = -\frac{x}{L} + \frac{2x^2}{L^2}$$

The assumed displacement distribution of the three-node element is more complicated than that of the two-node element. Thus, the three-node element can provide higher solution accuracy. However, the element requires more computational time because it contains more nodal unknowns.

The finite element equations for the two-node element are,

$$\frac{AE}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix}$$

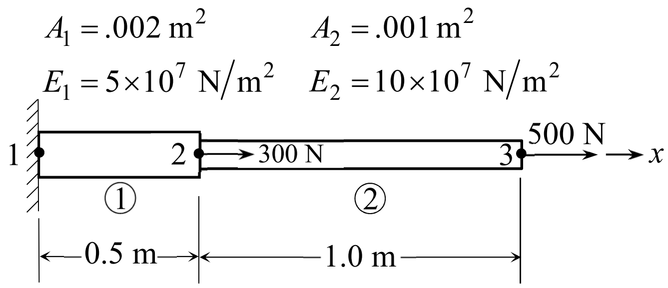
If we have a finite element model consisting of 10 elements, we need to establish 10 sets of finite element equations. These element equations are then assembled to form up a system of equations. The problem boundary conditions are applied before solving for the displacement unknowns at nodes.

If a finite element model containing many truss elements in two or three dimensions, the finite element equations above are needed to transform into to two or three dimensions accordingly. The transformation causes the finite element matrices to become larger leading to a larger set of algebraic equations. Such the larger set of algebraic equations requires more computer memory and computational time. However, these requirements do not pose any difficulty to current computers. Commercial software packages today have been developed to analyze complex truss structures containing a large number of elements effectively.

2.3 Academic Example

2.3.1 Two Truss Members in One Dimension

A model with two truss members connected together in one dimension is shown in the figure. The two truss members have the lengths of 0.5 and 1.0 m, cross-sectional areas of .002 and .001 m^2, and made from materials with Young's modulus of 5×10^7 and 10×10^7 N/m^2, respectively. The left end of the model is fixed at a wall while the connecting point and the right end are subjected to the forces of 300 and 500 N, respectively. By using only one two-node element to represent each truss member, determine the deformed configuration and the truss member stresses.



We will employ the Ansys Workbench to analyze this problem by going through the steps in details as follows.

(a) Starting Ansys Workbench

- Open the **Ansys Workbench**, set the **Units** menu on the upper tab to **Metric (kg,m,s,°C,A,N,V)**.
- On the **Analysis Systems** window, click twice on the **Static Structural** item. A new small box will appear on the **Project Schematic** window.
- Retype the name in the lower blue tab as the desired project name, e.g., **1D Truss Problem**, and hit **Enter**.
- Right click on the **Engineering Data** tab and select the **Edit…** option, the **A2: Engineering Data** window will appear. Double click on **Click here to add a new material** and type in a new material name, e.g., "*Material 1*", and hit **Enter**.
- Click at the **Isotropic Elasticity** under **Linear Elastic** and drag it to the **Property** list at the bottom of the window. Enter the **Young's Modulus** value as **5e7** and hit **Enter**,