Finite Element Method Crash Course with COMSOL

P. Dechaumphai

S. Sucharitpwatskul

Finite Element Method Crash Course with COMSOL

Finite Element Method Crash Course with COMSOL

Pramote Dechaumphai

Sedthawatt Sucharitpwatskul

Chulalongkorn University press

2023

430.-

Dechaumphai, Pramote
Finite element method crash course with COMSOL / Pramote Dechaumphai and Sedthawatt Sucharitpwatskul
1. Finite element method. 2. Finite element method -- Data processing. 3. Finite element method -- Computer programs. 4. COMSOL Multiphysics.
5. Engineering mathematics. I. Sucharitpwatskul, Sedthawatt.

620.00151825

ISBN (e-book) 978-974-03-4230-4

CUP 2627

cupress Knowledge to All
www.cupress.chula.ac.th

Published by **Chulalongkorn University Press**
First edition 2023
www.cupress.chula.ac.th [CUB6606-018E]
Tel. 0-2218-3562-3

Managing editor : Prof. Dr.Aran Hansuebsai Mrs.Orathai Nanthanadisai

Academic Editorial Department : Emeritus Professor Dr.Piyanart Bunnag
Assoc. Prof. Dr.Pimpan Dachakupt
Assoc. Prof.Chitsanu Pancharoen
Assoc. Prof. Dr.Vimolvan Pimpan

Coordinator : Wasana Sumsen

Proof reader : Treesunan Uparamai

Artwork and Cover : Chulalongkorn University Press
Contact : **Chulalongkorn University Book Center**
Phyathai Road, Pathumwan District, Bangkok 10330, Thailand
http://www.chulabook.com
Tel: 08-6323-3703-4
customer@cubook.chula.ac.th, info@cubook.chula.ac.th
Apps: CU-eBook Store

Preface

The book "Finite Element Method Crash Course with COMSOL" is written for beginners who want to understand the finite element method and be abele to use COMSOL Multiphysics software in a short time. The book contains 13 chapters in the fields of solid mechanics, heat transfer and fluid flows. In each chapter, the governing differential equations and finite element formulation are presented. An academic example is used to demonstrate the COMSOL software procedure for solving it in detail. An application example is included at the end of each chapter to highlight the software capability for analyzing realistic problems.

The last chapter highlights the software's ability to analyze various types of differential equations. The software is thus suitable for learning the finite element method to solve problems in many fields.

The authors would like to thank the COMSOL Inc through the i-Math Pte Ltd for providing the software to prepare this book. The authors also thank Chulalongkorn ©niversity Press for publishing and distributing the book.

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 to one dimensional problems 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:

Step 1: 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.

Step 2: 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.

Step 3: 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.

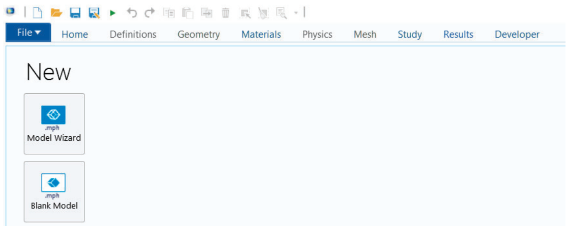
The six steps above indicate that the method is quite general and suitable for a large class of problems in science and engineering. The three problem ingredients which are the differential equations, boundary conditions and geometry are handled in the third, fifth and first step of the method, respectively.

1.3 COMSOL Multiphysics Software

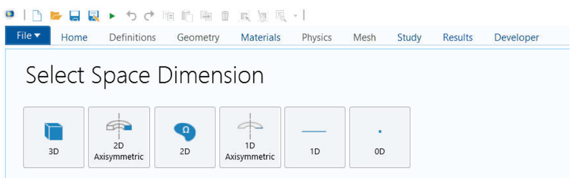
COMSOL is a cross-platform finite element analysis, solver and multiphysics simulation software. It was started in 1986 by Svante Littmarck and Farhad Saeidi at the Royal Institute of Technology in Stockholm, Sweden. Nowadays, COMSOL is widely used all over the world for analyzing a large class of science and engineering problems. These are electrical, structural, acoustic, fluid, heat transfer and chemical reaction problems, including their interaction. The software is easy to use and suitable for students who want to learn the finite element method. Problems can be solved effectively while solutions are displayed graphically on the computer screen.

Using the Graphic User Interface (GUI), we can interact with the software conveniently through mouse and keyboard. The analysis steps follow steps used in standard finite element method.

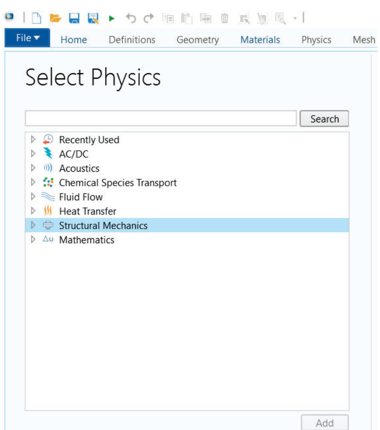
To analyze a problem, we start by clicking at the COMSOL logo, a **New** window appears as shown in the figure.

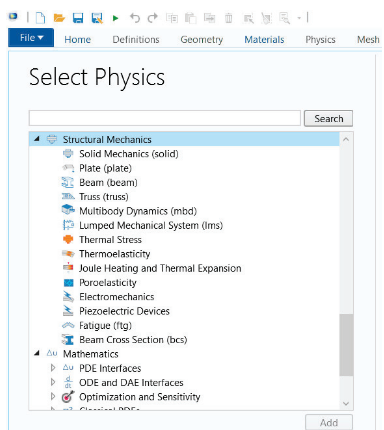


By clicking at the **Model Wizard** button, a **Select Space Dimension** window appears.

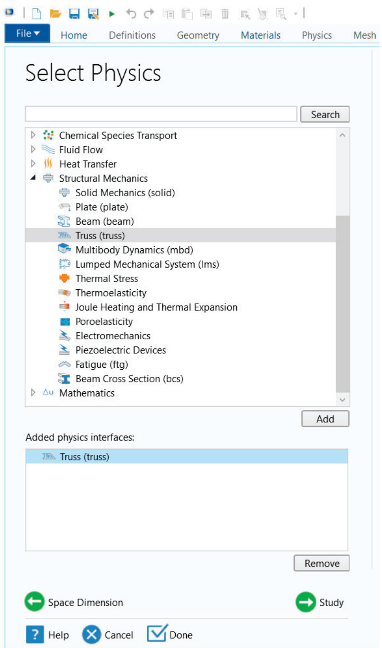


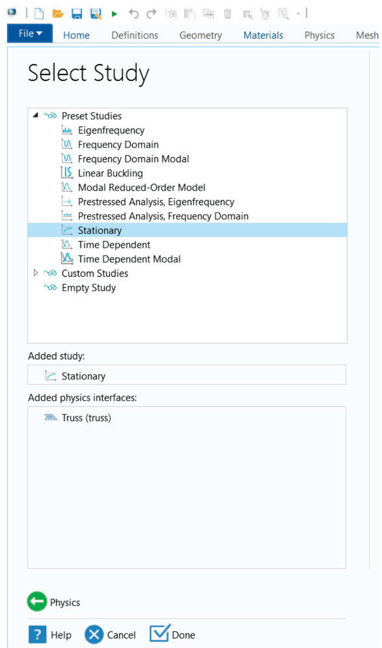
We then select the problem dimension that we will work with. For example, we click at **2D** button if we want to analyze a truss structure in two dimensions. By doing this, the **Select Physics** window appears as shown in the figure. We double click at the **Structural Mechanics** item, many analysis options pop up beneath it.



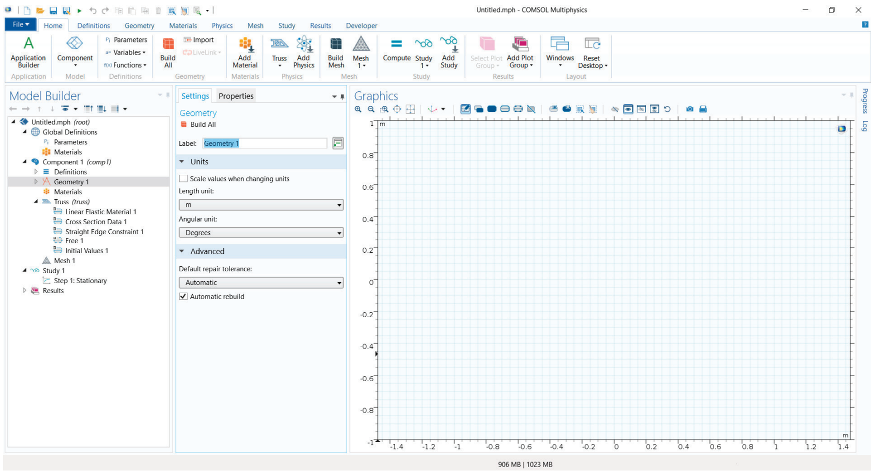


After selecting the **Truss (truss)** item beneath the **Structural Mechanics**, we click at the **Add** button. Then, click at **Study** button, a **Select Study** window appears. The **Select Study** window allows us to specify the type of study. As an example, if we want to perform static analysis of the truss problem, we select **Stationary** item as shown in the figure.





Finally, we click at **Done** button, a working window as shown in the figure appears. The working window consists of the command menu and large toolbars on the top. It also contains the three main windows which are the **Model Builder**, **Settings** and **Graphics** windows. We will use these windows to construct model geometries, apply boundary conditions, create meshes, solve for and display the solutions.



As mentioned earlier, the steps in using the software follow the standard finite element procedure. We will find that the software is easy to use by analyzing problems in many fields. These will be presented, step by step, in the following chapters.

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 analyses for structural, heat transfer and fluid flow problems, including problems in other engineering fields, will be presented in the following chapters with examples. Understanding materials in these chapters is encouraged before using the COMSOL 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,