

Finite Element Analysis

Within the various disciplines of engineering and science, Finite Element Analysis is becoming ever more widely used. Originally developed for aerospace structural analysis, FEA has grown to provide a convenient and speedy tool for approximation of the solution to a wide variety of complicated engineering problems. In the hands of a competent user, the method of FEA can produce accurate, reliable approximate solutions, at a small fraction of the cost of more rigorous, closed-form analyses. There exist any number of software packages for the performance of FEA, which are commercially available. Thus, basic knowledge of the background theory and knowledge of best practices of FEA is sufficient to allow an engineer to perform finite element analysis, once a software package is obtained and the engineer has time to learn its use.

This course provides the basic theoretical and practical knowledge (without unnecessary mathematic detail,) to allow an engineer to competently perform finite element analysis for

- Static structural analysis
- Steady-state thermal analysis
- Free vibrational analysis

using the FEA program ANSYS. In addition, the astute student can gather insight into performance of more complex analyses, such as transient dynamic analyses, contact, and fatigue, though these topics will not be specifically covered in this course. The course emphasizes theory only to the extent necessary for the user to understand the implications of FEA. It is not meant to produce finite element theorists, but rather a class of responsible users.

It is estimated that the course will, in total, ***require approximately 24 hours of work on the part of the student, over the duration of the six-week course.***

Computer Usage

The course will be delivered in online format. In addition, each student will receive on CD-ROM, an educational version of the FEA software ANSYS which will be used in this course. **Upon ANSYS approval the CD will be shipped - Please note that the approval and shipping process can take up to three weeks.**

The current version of ANSYS is Release 10.0 and includes both Classic and Workbench versions. The course utilizes the Classic version.

Finite Element Analysis Topics

The course has six modules, which cover topics as described in the following bullet points.

- **Introduction** - a brief overview of FEA, without mathematic examples.
Details of the method - a complete start-to-finish finite element analysis, demonstrating by example the steps of the method.
- **Introduction to ANSYS** - students get practical, hands-on experience performing FEA: a basic tutorial on the use of the commercially-available software. Students will work the example of the "details" module, and other tutorials for various analysis types.
- **More advanced topics in element generation** - introduction to concepts underlying the creation of "elements" which are used to make the approximation desired. The nuts and bolts of the method lie in element generation.
- **More ANSYS** - use of the finite element software for more advanced structural, thermal analyses, and basic modal analysis.
- **Good practices and common mistakes** - a description of various items of the method, which are helpful in making an analyst competent. How to model various boundary conditions, tips for reducing error, various other topics.