Method of Finite Elements I

- Lecturers: Prof. Dr. Eleni Chatzi, Dr. Patrick Steffen
- Assistants: Kostas Agathos (HIL E 19.6), Harry Mylonas (HIL H33.1), Konstantinos Tatsis (HIL H33.1)
- Course book: “Finite Element Procedures” by K.J. Bathe
Course Organization

• Lectures posted weekly on the class [webpage](#) (last year’s lecture slides also made available)

• Demos posted on the [Piazza platform](#) (all class members will receive an email to sign up to Piazza)

• Visiting hours: Prof. Dr. E. Chatzi (by email: chatzi@ibk.baug.ethz)

• TAs: every Thursday 13:00-15:00 in HIL E10.9. Preregistration at least 24 hours in advance required per Doodle. Posted on Piazza weekly.

• Further questions, will have to be submitted via the Piazza platform. You can also place anonymous Q&As if preferred.

• **Please ask questions! They help us better structure the class**
Course Evaluation

• Performance Assessment via submission of a Project.

• The Project will be divided into 3 parts.

• The full project will be posted on Piazza by 03 March 2018.

• Each part will be described according to the published schedule available on the webpage.

• The project will use part of own code in MATLAB (supported by the material offered in the demos), as well as comparison to a program of your choice (e.g: CUBUS, ABAQUS)

• Groups of 3 members are suggested. Contact us if you are having trouble finding a group.
Method of Finite Elements I

Project Submission

• **19.04.2018**: Midterm Project Update

• **24-25.05.2018 & 04-05.06.2018**: Final Project Submission - 15 min oral submission (printed slides & discussion)

• A Doodle will be distributed for signing up.

• **15.06.2018**: Deadline for Final report.
Method of Finite Elements I

A piece of Computational Mechanics

- Nanomechanics
- Micromechanics
- Continuum Mechanics
- Systems

Source: http://www.colorado.edu/MCEN/MCEN4173/chap_01.pdf
Method of Finite Elements I

A piece of Computational Mechanics

Structure  Microcracks  Pores  Pore solution

$REV_c$  $REV_{uc}$  $D^c$  $D^{uc} = D^{por}$  $D^{OF}$

Macro [cm]  Meso - Micro [μm - mm]  Nano [nm]
A piece of Computational Mechanics

- Nano-mechanics
- Micro-mechanics
- Continuum Mechanics
- Systems
- Solids & Structures
- Fluids
- Multiphysics
Method of Finite Elements I

A piece of Computational Mechanics

- Structural Mechanics
  - Statics
    - Time Invariant
  - Quasi-static
  - Dynamics
Method of Finite Elements I

A piece of Computational Mechanics

Structural Mechanics

Statics

Linear
Nonlinear

Dynamics

Linear
Nonlinear
Goals of the Class

- To present the linear FEM from a structural engineering perspective
- To introduce the concept of computation and link this to the analytical approach taught so far.
- To help you grasp the fundamental theory, and recognize how it is put in practice
- To learn how to put together simple analysis modules from scratch - then recognise this in the software you use.
- To understand how the software used in CE offices operate, and understand how to interpret results.
- To understand the associated challenges & limitations
Goals of the Class

• Remember! Software is continually changing but the core of the methods remains unchanged.
• It is the fundamental core that you need to master!

MATLAB will be used as a tool in this path to learning. It is however nothing but a tool, so feel free shift to any language you feel more comfortable with (if preferred).
Today’s lecture

- An overview of the MFE I course
- MFE: A historical overview
- Introduction to the use of Finite Elements
  - Why and when to use FEM?
  - Challenges for a correct use/design
- FEM into Practice
# Course Overview

<table>
<thead>
<tr>
<th>Date</th>
<th>Lecture</th>
<th>Project</th>
</tr>
</thead>
<tbody>
<tr>
<td>19.02</td>
<td>Lecture 1: Introduction</td>
<td></td>
</tr>
<tr>
<td>26.02</td>
<td>Lecture 2: The Direct Stiffness Method</td>
<td></td>
</tr>
<tr>
<td>02.03</td>
<td>Project posted on Piazza</td>
<td></td>
</tr>
<tr>
<td>05.03</td>
<td>Lecture 3: Stability &amp; Modal Analysis</td>
<td>Project Part I description</td>
</tr>
<tr>
<td>12.03</td>
<td>Computer Lab I</td>
<td></td>
</tr>
<tr>
<td>19.03</td>
<td>Lecture 4: The Variational Formulation</td>
<td></td>
</tr>
<tr>
<td>26.03</td>
<td>Lecture 5: The Isoparametric formulation</td>
<td>Project Part II description</td>
</tr>
<tr>
<td>09.04</td>
<td>Lecture 6: Prismatic elements</td>
<td></td>
</tr>
</tbody>
</table>

*lecture sessions come with a Demo in the last part of the session*
# Course Overview

<table>
<thead>
<tr>
<th>Date</th>
<th>Lecture</th>
<th>Project</th>
</tr>
</thead>
<tbody>
<tr>
<td>19.04</td>
<td>Midterm Project Update (during TA weekly hours)</td>
<td></td>
</tr>
<tr>
<td>23.04</td>
<td>Lecture 7: 2D Elements/The Plate Element</td>
<td>Project Part III description</td>
</tr>
<tr>
<td>30.04</td>
<td>Lecture 8: Practical application of the MFE - Practical Considerations</td>
<td></td>
</tr>
<tr>
<td>07.05</td>
<td>Lecture 9: Practical application of the MFE- Results Interpretation</td>
<td></td>
</tr>
<tr>
<td>14.05</td>
<td>Computer Lab II</td>
<td></td>
</tr>
<tr>
<td>24-25.05 &amp; 04-05.06</td>
<td>Project Submission (15 min sessions)</td>
<td></td>
</tr>
<tr>
<td>28.05</td>
<td>no class</td>
<td></td>
</tr>
</tbody>
</table>

*lecture sessions come with a Demo in the last part of the session*
Chapter 1: FE Analysis in brief…

The goal of Structural Analysis: Modeling a system by means of approximation.

How to do this?
Come up with a simplified idealization. This may oftentimes be achieved conceptually.

Example
Chapter 1: FE Analysis in brief...

The goal of Structural Analysis: Modeling a system by means of approximation.

How to do this?
Come up with a simplified idealization. This may oftentimes be achieved conceptually.

Example
How to model this?
Chapter 1: FE Analysis in brief…

The goal of Structural Analysis: Modeling a system by means of approximation.

How to do this?
Come up with a simplified idealization. This may oftentimes be achieved conceptually.

Example
How to model this?

Break the system down to components.
Can we still model this by hand? Perhaps…
Chapter 1: FE Analysis in brief...

The goal of Structural Analysis: Modeling a system by means of approximation.

How to do this?
Come up with a simplified idealization. This may oftentimes be achieved conceptually.

Example
What if we want to model a critical component of this system?
Can we still model this by hand?
The task is now harder.
Chapter 1: FE Analysis in brief…

How to approximate complex structures/domains?

Numerical Methods

- Finite Difference Method
- Finite Element Method
- Boundary Element Method

discretization error
Chapter 1: FE Analysis in brief…

FEA was originally developed for solid mechanics applications.

**Object:** A Solid with known mechanical properties. (a skyscraper; a shaft; bio tissue …)

**Main Features**

- *Acting Loads*
- *Boundary:* The surface enclosing the geometry
- *Solid:* Interior + Boundary
- *Boundary conditions:* prescribed displacements/tractors on the boundary
FE Analysis in brief...

Problem Statement

undeformed

? 

dehformed
How can we use FEM to solve this problem?

The Finite Element Method (FEM) is a numerical method for solution of systems where the governing equations are expressed in the form of partial differential equations (PDEs).

It essentially degenerated the original PDE problem to a system of algebraic equations which are solvable via traditional linear algebra techniques.

Simply put, FEM is a method for breaking down a complex problem/domain into smaller elements that are interconnected and assembled to form an approximation to the original structure.
FE Analysis in brief…

Simply put, FEM is a method for breaking down a complex problem/domain into smaller elements that are interconnected and assembled to form an approximation to the original structure.

Does this ring a bell?

Baustatik II – The Direct Stiffness Method
MFE Historical Evolution

How was FEM historically formed? (source: Carlos A. Felippa)

The MFE is the confluence of three ingredients: matrix structural analysis, variational approach and a computer.

Theoretical Formulation

1. “Lösung von Variationsproblemen” by W. Ritz in 1908
2. “Weak formulation” by B. Galerkin in 1915
3. “Mathematical foundation” by R. Courant ca. 1943
MFE Historical Evolution

How was FEM historically formed?

Who can be mostly blamed for the reign of FEM?

1950s, M.J. Turner at Boeing (aerospace industry in general): Direct Stiffness Method

“Turner got Boeing to commit resources to it while other aerospace companies (Douglas, Lockheed, Rockwell, ...) were mired in the Force Method swamp. He oversaw the development of the first continuum based finite elements, the consequent creation and expansion of the DSM, and its first application to nonlinear problems.

In addition to Turner, major contributors to current practice include: B. M. Irons, inventor of isoparametric models, shape functions, the patch test and frontal solvers; R. J. Melosh, who recognized the Rayleigh-Ritz link and systematized the variational derivation of stiffness elements; and E. L. Wilson, who developed the first open source (and widely imitated and distributed) FEM and matrix software”

(source: Carlos A. Felippa)
MFE Historical Evolution

How was FEM historically formed?

Who were the “populizers”?

1. Matrix formulation of structural analysis by Agyris in 1954

“As a consultant to Boeing in the early 1950s, Argyris, a Force Method expert then at Imperial College, received reports from Turner’s group, and weaved the material into the above influential publication. He was the first to construct a displacement-assumed continuum element.”

2. The term ‘Finite Element‘ was coined by Clough in 1960

“R.W. Clough and H.C. Martin, then junior professors at U.C. Berkeley and U. Washington, respectively, spent “faculty internship” summers at Turner’s group during 1952–53. The result of this seminal collaboration was a celebrated paper, widely considered the start of the present FEM.”

3. First book on FEM by Zienkiewicz and Cheung in 1967

“Olek Zienkiewicz, originally an expert in finite difference methods was convinced in 1958 by Clough to try FEM. He went on to write the first textbook on the subject”
MFE Development

Commercial Software Development

1. General purpose packages for main frames in 1970s
2. Special purpose software for PCs in 1980s

During this class, the following software packages will be used: ABAQUS, CUBUS, as well as code snippets in MATLAB
FEM is a big success story, because it…

1. can handle complex geometries
2. can handle a wide variety of engineering problems
   - mechanics of solids & fluids
   - dynamics/heat/electrostatic problems…
3. can handle complex restraints & loading
4. is very well suited for computation
What is a Finite Element?

(a)

Find the perimeter $L$ of a circle of diameter $d$
We know that $L = \pi d$.

Approximation via Inscribed Polygon
The element length is $L_{ij} = 2r \sin(\pi/n)$.
Since all elements have the same length, the polygon perimeter is $L_n = n L_{ij}$
Hence the approximation to $\pi$ is $\pi n = L_n/d = n \sin(\pi/n)$.

Source: http://www.colorado.edu/MCEN/MCEN4173/chap_01.pdf
What is a Finite Element?

(a) A circle with radius $r$ and diameter $d$.

(b) An inscribed regular polygon with $n$ sides.

(c) A simple finite element.

(d) An element with nodes $i$ and $j$.

Rectification of Circle by Inscribed Polygons ("Archimedes FEM")

<table>
<thead>
<tr>
<th>$n$</th>
<th>$\pi_n = n \sin(\pi/n)$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.000000000000000000</td>
</tr>
<tr>
<td>2</td>
<td>2.000000000000000000</td>
</tr>
<tr>
<td>4</td>
<td>2.828427124746190000</td>
</tr>
<tr>
<td>8</td>
<td>3.061467458920718000</td>
</tr>
<tr>
<td>16</td>
<td>3.121445152258052000</td>
</tr>
<tr>
<td>32</td>
<td>3.136548490545939000</td>
</tr>
<tr>
<td>64</td>
<td>3.140331156954753000</td>
</tr>
<tr>
<td>128</td>
<td>3.141277250932773000</td>
</tr>
<tr>
<td>256</td>
<td>3.141513801144301000</td>
</tr>
</tbody>
</table>

Source: [http://www.colorado.edu/MCEN/MCEN4173/chap_01.pdf](http://www.colorado.edu/MCEN/MCEN4173/chap_01.pdf)
Classification of Engineering Systems

Discrete

\[ F = KX \]

Direct Stiffness Method

Continuous

\[ k \left( \frac{\partial^2 \phi}{\partial^2 x} + \frac{\partial^2 \phi}{\partial^2 y} \right) = 0 \]

Laplace Equation

**FEM:** Numerical Technique for **approximating** the solution of **continuous systems**. We will use a displacement based formulation and a stiffness based solution (direct stiffness method).
How does it work?

• Within the framework of continuum mechanics dependencies between geometrical and physical quantities are formulated on a differentially small element and then extended to the whole continuum.

• As a result we obtain differential, partial differential or integral equations for which, generally, an analytical solution is not available – they have to be solved using some numerical procedure.

• The MFE is based on the physical discretization of the observed domain, thus reducing the number of the degrees of freedom; moreover the governing equations are, in general, algebraic.
Why & When to use FEM?

Let’s follow Clough’s trip:

“When I arrived in the summer of 1952, Jon Turner asked me to work on the vibration analysis of a delta wing structure. Because of its triangular plane form, this problem could not be solved by procedures based on standard beam theory; so I spent the summer of 1952 trying to formulate a delta wing model built up of an assembly of one-dimensional beams and struts. However, the results of deflection analysis based on this type of mathematical model were in very poor agreement with data obtained from laboratory tests of a scale model of a delta wing. My final conclusion was that my summer’s work was a total failure — however, at least I learned what did not work.

Spurred by this disappointment, I decided to return to Boeing for the summer faculty program of 1953. During the winter, I stayed in touch with Jon Turner so I was able to rejoin the structural dynamics unit in June.”

(source: Carlos A. Felippa)
Why & When to use FEM?

Let’s follow Clough’s trip: (source: Carlos A. Felippa)

“The most important development during the winter was that Jon suggested we try to formulate the stiffness properties of the wing by assembling plane stress plates of either triangular or rectangular shapes. So I developed stiffness matrices for plates of both shapes, but I decided the triangular form was much more useful because such plates could be assembled to approximate structures of any configuration. Moreover, the stiffness properties of the individual triangular shapes could be calculated easily based on assumptions of uniform states of normal stress in the X and Y directions combined with a uniform state of shear stress. Then the stiffness of the complete structure was obtained by appropriate addition from the contributions of the individual pieces.

The remainder of the summer of 1953 was spent in demonstrating that deflections calculated for structures formed of assemblies of triangular elements agreed well with laboratory measurements on the actual physical models.”
How does it work?

Physical Model
Describe the problem: Simplify a real engineering problem into a problem that can be solved by FEA

FE model
Discretize/mesh the solid, define material properties, apply boundary conditions

Theory
Choose approximate functions, formulate linear equations, and solve equations

Results
Obtain, visualize and explain the results

Post-processor

Source: http://www.colorado.edu/MCEN/MCEN4173/chap_01.pdf
Step 1: Problem Formulation

The formulation of the equations governing the response of a system under specific loads and constraints at its boundaries is usually provided in the form of a differential equation. The differential equation also known as the strong form of the problem is typically extracted using the following sets of equations:

1. **Equilibrium Equations**
   
   \[ f(x) = R + \frac{aL + ax}{2} (L - x) \]

2. ** Constitutive Requirements Equations**
   
   \[ \sigma = E \varepsilon \]

3. **Kinematics Relationships**
   
   \[ \varepsilon = \frac{du}{dx} \]
Steps in the MFE

Step 2: Discretization/Meshing

The continuum is discretized using a mesh of finite elements.
Steps in the MFE

**Discretization:** The continuum is discretized using a mesh of finite elements.

These elements are connected at nodes located on the element boundaries.
Steps in the MFE

Step 3: Interpolation - The Shape Functions

The state of deformation, stresses, etc. in each element is approximated by a set of corresponding values in the nodes; these nodal values are the basic unknowns of the MFE.

Values between nodes, have to therefore result via interpolation.

- Bounded and Continuous
- One for each node
- \( N_i^e(x_j^e) = \delta_{ij} \), where

\[
\delta_{ij} = \begin{cases} 
1 & \text{if } i = j \\
0 & \text{if } i \neq j
\end{cases}
\]

The way in which these steps are approached has a great influence on the results of the calculations.
Challenges

• The MFE is only a way of solving the mathematical model
• The solution of the physical problem depends on the quality of the mathematical model – the choice of the mathematical model is crucial
• The chosen mathematical model is reliable if the required response can be predicted within a given level of accuracy compared to the response of a very comprehensive (highly refined) mathematical model
• The most effective mathematical model for the analysis is the one that gives the required response with sufficient accuracy and at the lowest computational toll
Simple Example

Complex physical problem modelled by a simple mathematical model

\[ W = 1000 \text{ N} \]
\[ L = 27.5 \text{ cm} \]
\[ r_N = 0.5 \text{ cm} \]
\[ E = 2 \times 10^7 \text{ N/cm}^2 \]
\[ v = 0.3 \]
\[ h = 6.0 \text{ cm} \]
\[ t = 0.4 \text{ cm} \]

\[ M = WL \]
\[ = 27,500 \text{ N cm} \]

\[ \delta_{\text{at load}} = \frac{1}{3} \frac{W(L + r_N)^3}{EI} + \frac{W(L + r_N)}{AG} \]
\[ = 0.053 \text{ cm} \]
Simple Example

Equilibrium equations (see Example 4.2)

\[
\begin{align*}
\frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} &= 0 \\
\frac{\partial \tau_{yx}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} &= 0
\end{align*}
\]

in domain of bracket

\(\tau_{nn} = 0, \tau_{nt} = 0\) on surfaces except at point B
and at imposed zero displacements

Stress-strain relation (see Table 4.3):

\[
\begin{bmatrix}
\tau_{xx} \\
\tau_{yy} \\
\tau_{xy}
\end{bmatrix}
= \frac{E}{1 - \nu^2}
\begin{bmatrix}
1 & \nu & 0 \\
\nu & 1 & 0 \\
0 & 0 & (1 - \nu)/2
\end{bmatrix}
\begin{bmatrix}
\varepsilon_{xx} \\
\varepsilon_{yy} \\
\gamma_{xy}
\end{bmatrix}
\]

\(E = \) Young’s modulus, \(\nu = \) Poisson’s ratio

Strain-displacement relations (see Section 4.2):

\[
\begin{align*}
\varepsilon_{xx} &= \frac{\partial u}{\partial x}; & \varepsilon_{yy} &= \frac{\partial v}{\partial y}; & \gamma_{xy} &= \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}
\end{align*}
\]

\(\delta_{\text{at load w}} = 0.064 \text{ cm}\)

\(M|_{x=0} = 27,500 \text{ N cm}\)

Detailed reference model – 2D plane stress model
Considerations

• Choice of mathematical model must correspond to desired response

• The most effective mathematical model delivers reliable answers with the least amount of effort

• Any solution (including MFE) of a mathematical model is limited to information contained in or fed into the model: bad input – bad output (garbage in – garbage out)

• Assessment of accuracy is based on comparisons with the results from very comprehensive models – but in practice it has to be based on experience (experiments…)

• The engineer (user) should be able to judge the quality of the obtained results (i.e. for plausibility)