EK2360 - Hands-on Micro-Electromechanical Systems Engineering

Introduction to FEM Modelling with COMSOL

Bernhard Beuerle
Fritzi Töpfer

Micro and Nanosystems
School of Electrical Engineering
KTH Royal Institute of Technology

November 5, 2015

Table of Contents

Introduction to the Finite Element Method (FEM)

Introduction to COMSOL Multiphysics

Tutorial on COMSOL Multiphysics

Table of Contents

Introduction to the Finite Element Method (FEM)

Introduction to COMSOL Multiphysics

Tutorial on COMSOL Multiphysics

What is FEM?

The FEM is a numerical method seeking an approximated solution of the distribution of field variables in the problem domain.

What is FEM?

- solves boundary value problems/field problems
- finds the distribution of field variables (= the dependent variables of interest governed by the differential equations)

Examples - stress analysis

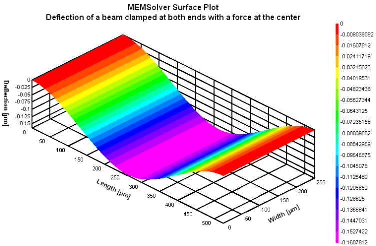


Figure: displacement field

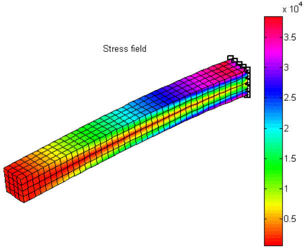


Figure: stress field

Examples - electrical analysis

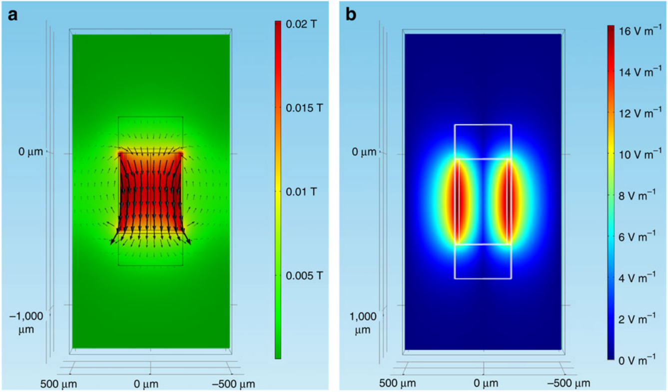


Figure: magnetic and electric field distributions

Examples - electrical analysis

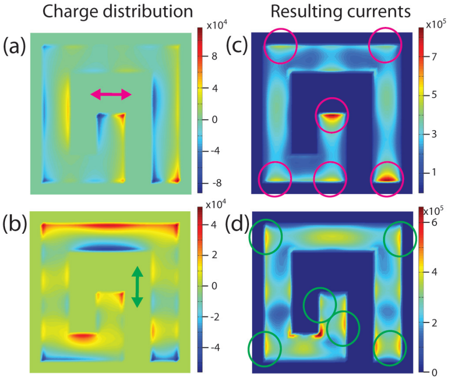


Figure: charge and current distributions

Examples - thermal analysis

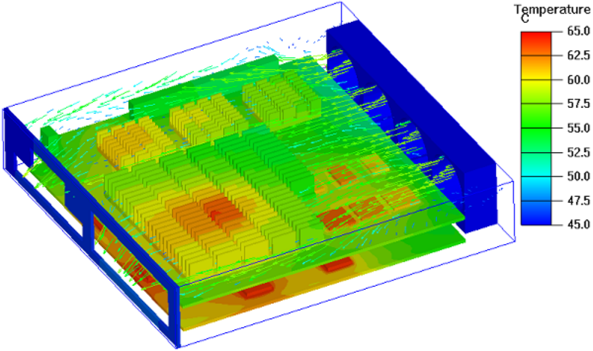
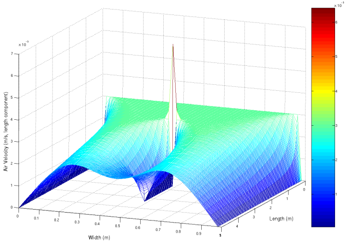
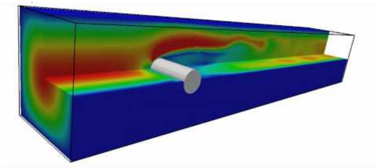


Figure: temperature distribution

Examples - fluid flow analysis

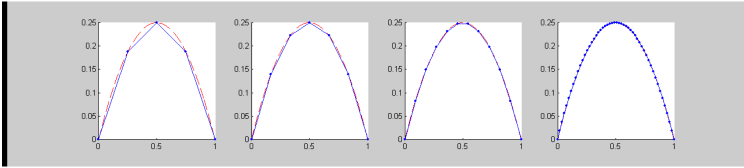




Navier–Stokes differential equations used to simulate airflow around an obstruction

FEM

- is a numerical method
- finds an *approximated* solution



Why use FEM?

The behavior of a phenomenon in a system depends on:

- geometry / domain
- material properties
- initial, boundary and loading conditions

$\rightarrow$ can be very complex

$\rightarrow$ difficult to solve governing differential equations by analytical means

$\Rightarrow$ Therefore: use numerical method to find approximated solution.

How does the FEM work?

1. dividing the problem domain into several elements
2. known physical laws are then applied to each small element, each of which usually has a very simple geometry
3. proper principles are followed to establish equations for the elements
4. the elements are *tied* to one another
5. this process leads to a set of linear algebraic simultaneous equations for the entire system that can be solve easily to yield the required field variable

Advantages of FEM

- can readily handle very complex geometries
  - the heart and power of the FEM
- can handle a wide variety of engineering problems
  - solid mechanics - dynamics - heat problems
  - fluids - electrostatic problems
- can handle complex loading
  - nodal load (point loads)
  - element load (pressure, thermal, inertial forces)
  - time or frequency dependent loading

Disadvantages of FEM

- a general **closed-form solution**, which would permit one to examine system response to changes in various parameters, **is not produced**
- the FEM obtains only *approximated* solutions
- the FEM has *inherent* errors
- mistakes by users can be fatal

Table of Contents

Introduction to the Finite Element Method (FEM)

Introduction to COMSOL Multiphysics

Tutorial on COMSOL Multiphysics

Introduction to COMSOL Multiphysics

- model and simulate any physics-based system
- includes the COMSOL Desktop graphical user interface (GUI) and a set of physics interface (= predefined user interfaces with associated modeling tools)
- suitable for multiphysics problems

$\Rightarrow$ COMSOL Multiphysics is a well filled tool box for solving PDEs in an approximate way using the FEM

General workflow

1. Geometry
2. PDE definition
   - Subdomain settings)
   - Boundary, Initial and Loading conditions
   - Material property
3. Meshing (Discretization)
   - Moving mesh
4. PDE solution
   - Different solvers
5. Post-processing

Table of Contents

Introduction to the Finite Element Method (FEM)

Introduction to COMSOL Multiphysics

Tutorial on COMSOL Multiphysics

Using COMSOL Multiphysics

Calculate the spring constant of a basic cantilever.

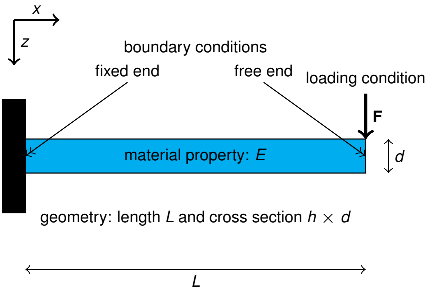
COMSOL Multiphysics demonstration

Finite Element Modeling

1. Geometry
2. PDE definition
3. Meshing (Discretization)
4. PDE solution
5. Post-processing

EULER–BERNOULLI equation: $\frac{\mathrm{d}^2}{\mathrm{d}x^2}\left(EI\frac{\mathrm{d}^2 w}{\mathrm{d}x^2}\right) = q$

$$I = \iint z^2 \, dy \, dz$$



- displacement of cantilever?
- spring constant?