16.810

Engineering Design and Rapid Prototyping

CAE -Finite Element Method

Instructor(s)

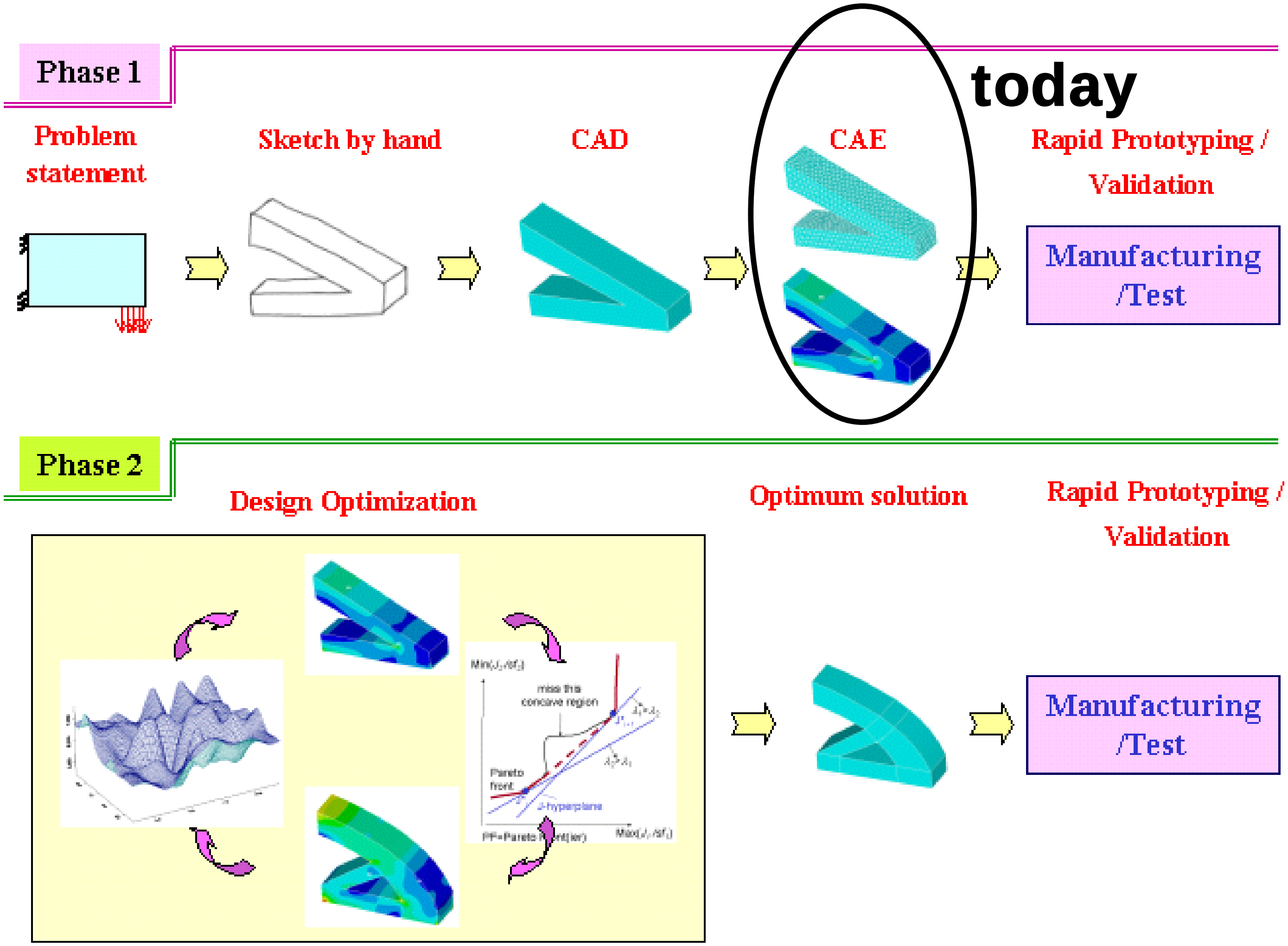
Prof. Olivier de Weck

January 11, 2005

16.810 Plan for Today

- Hand Calculations Aero → Structures
- FEM Lecture (ca. 45 min)
  - FEM fundamental concepts, analysis procedure
  - Errors, Mistakes, and Accuracy
- Cosmos Introduction (ca. 30 min)
  - Given by TA
  - Follow along step-by-step
- Work on CAD and conduct FEA in teams(ca. 90 min)
  - Work in teams of two
  - First conduct an analysis of your CAD design
  - You are free to make modifications to your original model

Course Concept



Phase 1

today

Problem statement

Sketch by hand

CAD

CAE

Rapid Prototyping / Validation

Manufacturing /Test

Phase 2

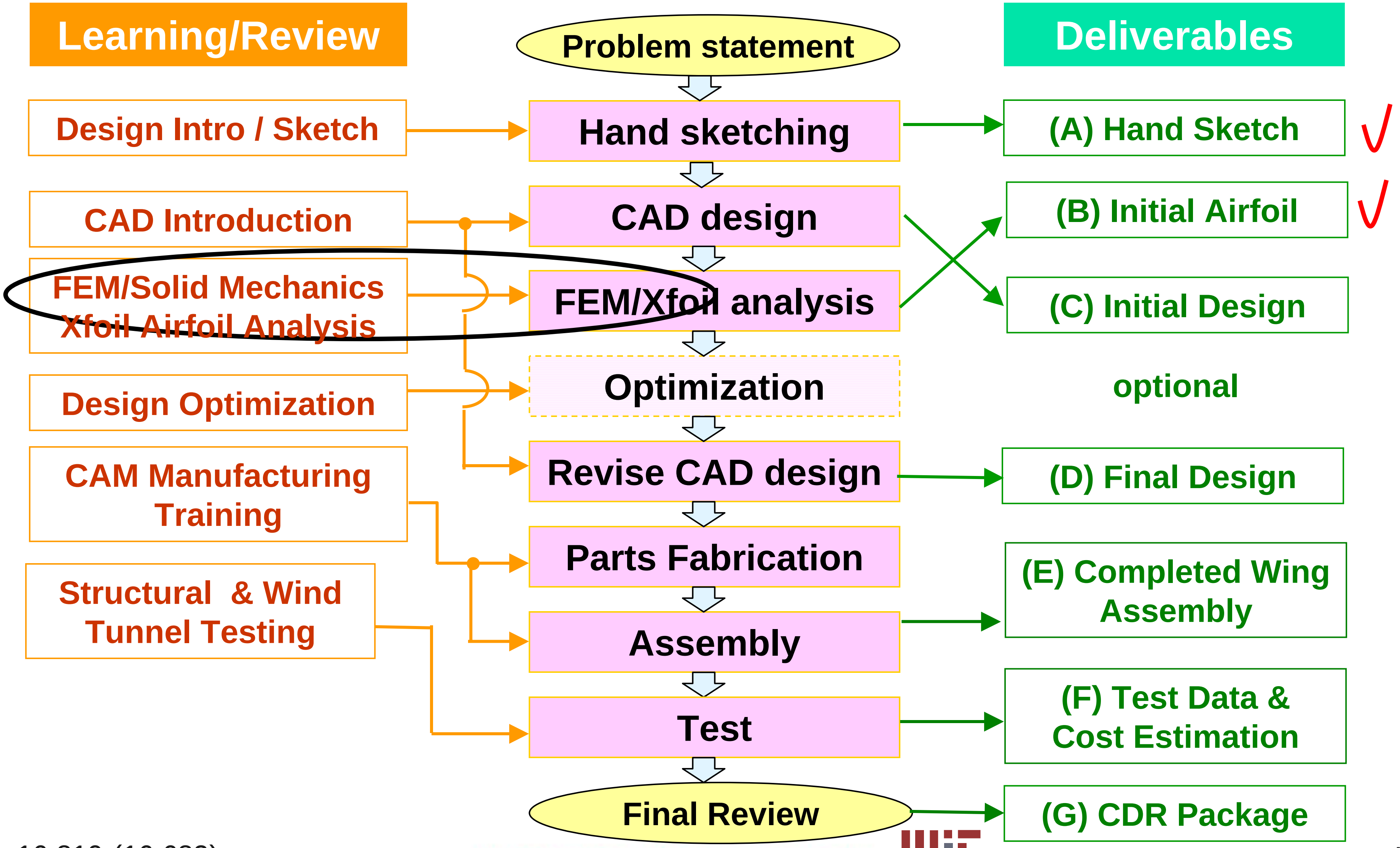
Design Optimization

Optimum solution

Rapid Prototyping / Validation

Manufacturing /Test

Course Flow Diagram (2005)



Learning/Review

- Design Intro / Sketch
- CAD Introduction
- FEM/Solid Mechanics Xfoil Airfoil Analysis
- Design Optimization
- CAM Manufacturing Training
- Structural & Wind Tunnel Testing

Problem statement

- Hand sketching
- CAD design
- FEM/Xfoil analysis
- Optimization
- Revise CAD design
- Parts Fabrication
- Assembly
- Test

Final Review

Deliverables

- (A) Hand Sketch ✓
- (B) Initial Airfoil ✓
- (C) Initial Design
- optional
- (D) Final Design
- (E) Completed Wing Assembly
- (F) Test Data & Cost Estimation
- (G) CDR Package

Numerical Method

Finite Element Method

Boundary Element Method

Finite Difference Method

Finite Volume Method

Meshless Method

What is the FEM?

FEM: Method for numerical solution of field problems.

Description

- **FEM cuts a structure into several elements (pieces of the structure).**
- **Then reconnects elements at “nodes” as if nodes were pins or drops of glue that hold elements together.**
- **This process results in a set of simultaneous algebraic equations.**

Number of degrees-of-freedom (DOF)

Continuum: **Infinite**

FEM: **Finite**

(This is the origin of the name, Finite Element Method)

Fundamental Concepts (1)

Many engineering phenomena can be expressed by "governing equations" and "boundary conditions"

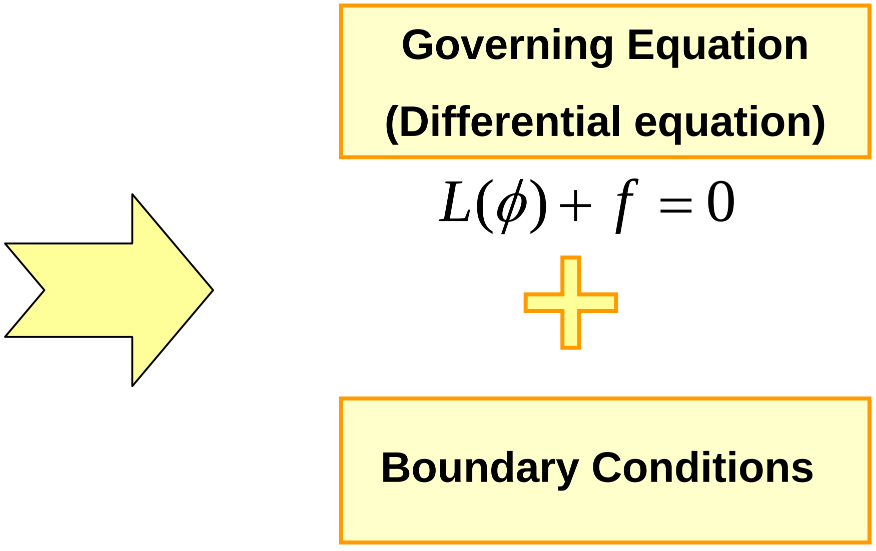
Elastic problems

Thermal problems

Fluid flow

Electrostatics

etc.



Governing Equation
(Differential equation)

$$L(\phi) + f = 0$$

+

Boundary Conditions

$$B(\phi) + g = 0$$

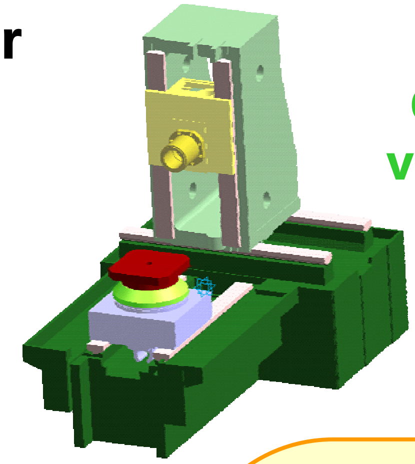
Fundamental Concepts (2)

Example: Vertical machining center

Elastic deformation

Thermal behavior

etc.



Geometry is very complex!

Governing Equation: $L(\phi) + f = 0$

Boundary Conditions: $B(\phi) + g = 0$

You know all the equations, but you cannot solve it by hand

FEM

Approximate!

A set of simultaneous algebraic equations

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

Fundamental Concepts (3)

$$[\mathbf{K}]\{\mathbf{u}\} = \{\mathbf{F}\} \quad ⇨ \quad \{\mathbf{u}\} = [\mathbf{K}]^{-1}\{\mathbf{F}\}$$

Property

Behavior

Action

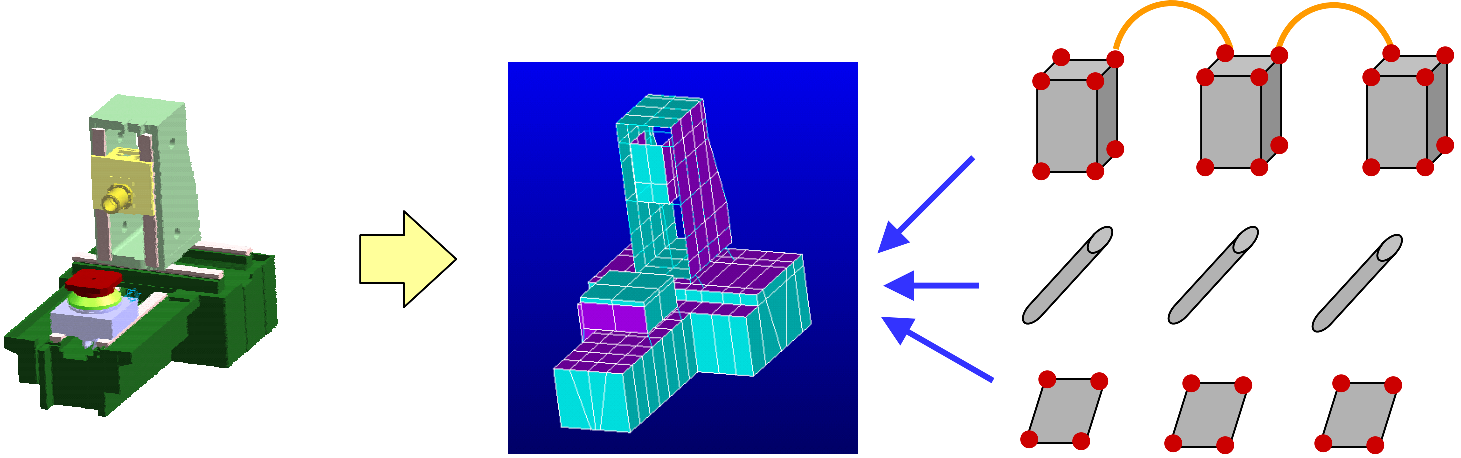
Unknown

	Property $[\mathbf{K}]$	**Behavior** $\{\mathbf{u}\}$	**Action** $\{\mathbf{F}\}$
Elastic	stiffness	displacement	force
Thermal	conductivity	temperature	heat source
Fluid	viscosity	velocity	body force
Electrostatic	Dielectric permittivity	electric potential	charge

Fundamental Concepts (4)

It is very difficult to solve the algebraic equations for the entire domain

- Divide the domain into a number of small, simple elements
  - A field quantity is interpolated by a polynomial over an element
    - Adjacent elements share the DOF at connecting nodes

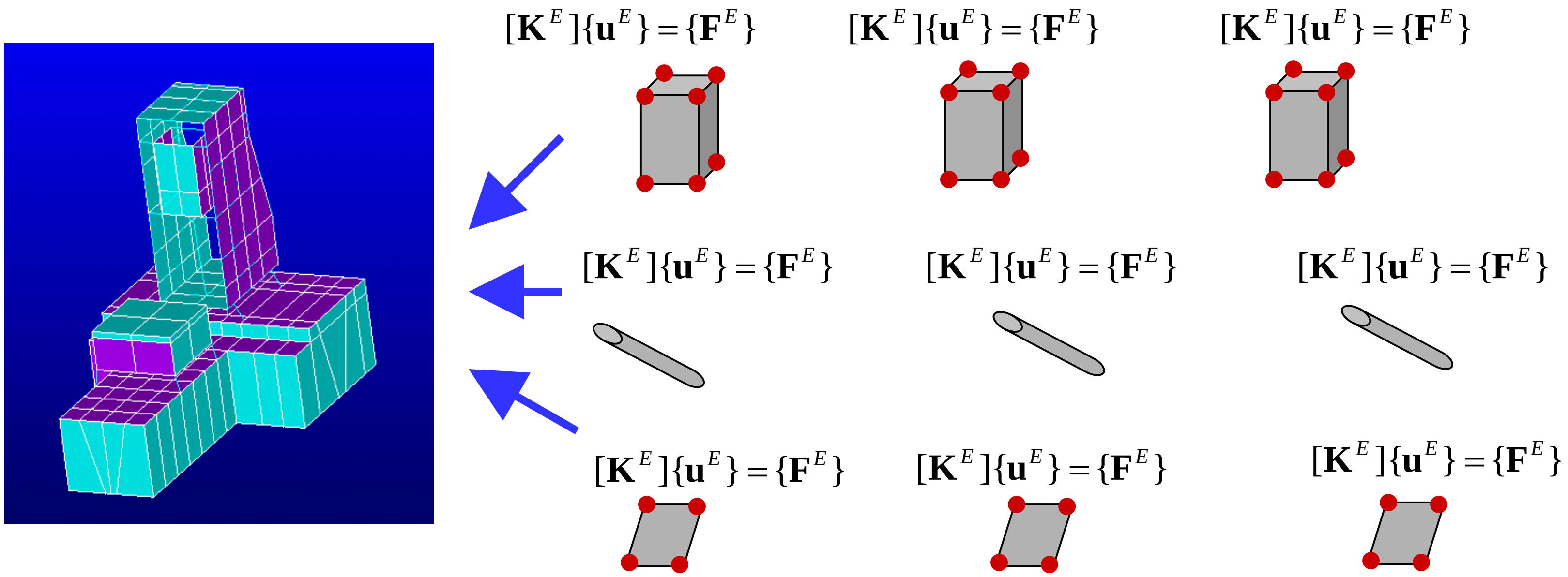


Finite element: Small piece of structure

Fundamental Concepts (5)

Obtain the algebraic equations for each element (this is easy!)

➡ **Put all the element equations together**



$[\mathbf{K}^E]\{\mathbf{u}^E\}=\{\mathbf{F}^E\}$ $[\mathbf{K}^E]\{\mathbf{u}^E\}=\{\mathbf{F}^E\}$ $[\mathbf{K}^E]\{\mathbf{u}^E\}=\{\mathbf{F}^E\}$

$[\mathbf{K}^E]\{\mathbf{u}^E\}=\{\mathbf{F}^E\}$ $[\mathbf{K}^E]\{\mathbf{u}^E\}=\{\mathbf{F}^E\}$ $[\mathbf{K}^E]\{\mathbf{u}^E\}=\{\mathbf{F}^E\}$

$[\mathbf{K}^E]\{\mathbf{u}^E\}=\{\mathbf{F}^E\}$ $[\mathbf{K}^E]\{\mathbf{u}^E\}=\{\mathbf{F}^E\}$ $[\mathbf{K}^E]\{\mathbf{u}^E\}=\{\mathbf{F}^E\}$

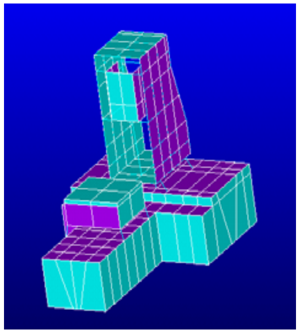
$$[\mathbf{K}]\{\mathbf{u}\}=\{\mathbf{F}\}$$

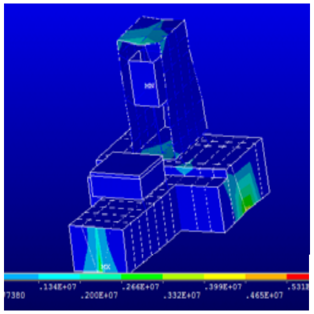
Fundamental Concepts (6)

Solve the equations, obtaining unknown variabless at nodes.

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

$$\{\mathbf{u}\} = [\mathbf{K}]^{-1}\{\mathbf{F}\}$$



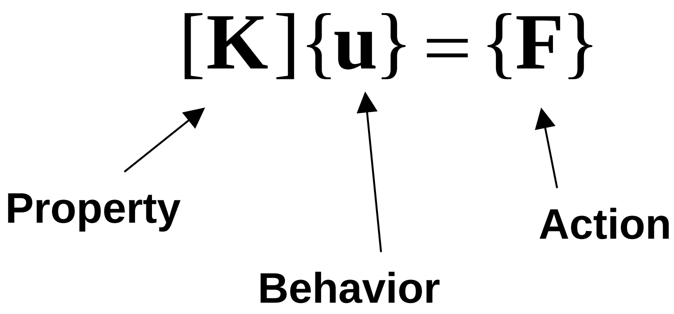


Concepts - Summary

- FEM uses the concept of piecewise polynomial interpolation.

- By connecting elements together, the field quantity becomes interpolated over the entire structure in piecewise fashion.

- A set of simultaneous algebraic equations at nodes.



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

Property

Behavior

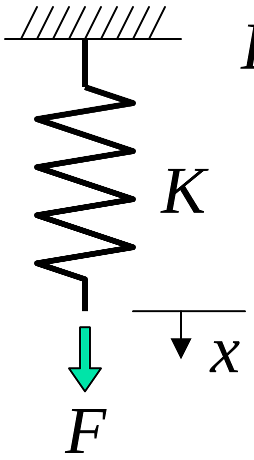
Action

$$Kx = F$$

K: Stiffness matrix

x: Displacement

F: Load



Brief History

- The term finite element was first coined by Clough in 1960. In the early 1960s, engineers used the method for approximate solutions of problems in stress analysis, fluid flow, heat transfer, and other areas.

- The first book on the FEM by Zienkiewicz and Chung was published in 1967.

- In the late 1960s and early 1970s, the FEM was applied to a wide variety of engineering problems.

- Most commercial FEM software packages originated in the 1970s. (Abaqus, Adina, Ansys, etc.)

- Klaus-Jurgen Bathe in ME at MIT

Reference [2]

Advantages of the FEM

Can readily handle very complex geometry:

- 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 restraints

- Indeterminate structures can be solved.

Can handle complex loading

- Nodal load (point loads)

- Element loads - distributed (pressure, thermal, inertial forces)

- Time or frequency dependent loading

Disadvantages of the 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 "approximate" solutions.

The FEM has "inherent" errors.

Mistakes by users can remain undetected.

Typical FEA Procedure by Commercial Software

User **Preprocess** Build a FE model

Computer **Process** Conduct numerical analysis

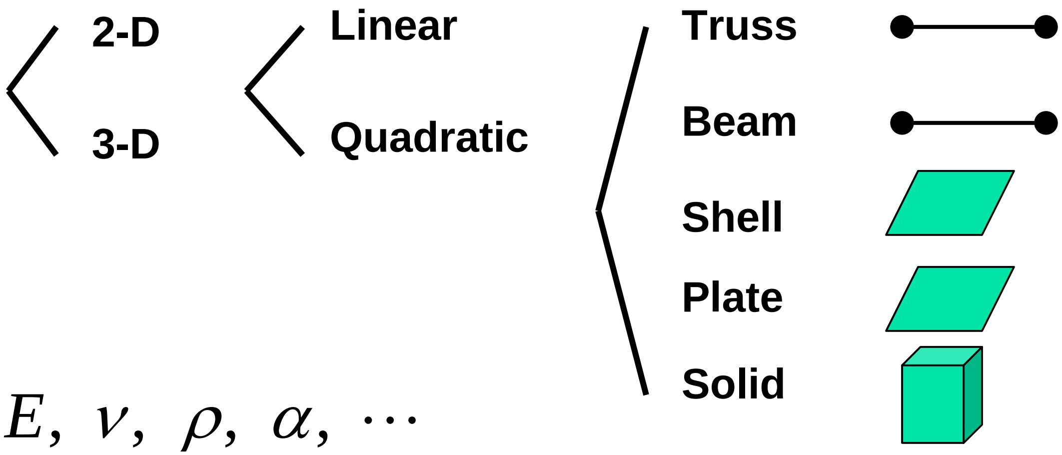
User **Postprocess** See results

Preprocess (1)

[1] Select analysis type

- Structural Static Analysis
- Modal Analysis
- Transient Dynamic Analysis
- Buckling Analysis
- Contact
- Steady-state Thermal Analysis
- Transient Thermal Analysis

[2] Select element type



2-D
3-D

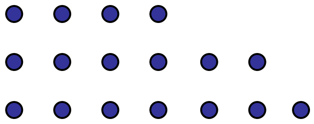
Linear
Quadratic

Truss
Beam
Shell
Plate
Solid

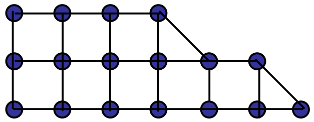
[3] Material properties $E,\ \nu,\ \rho,\ \alpha,\ \cdots$

Preprocess (2)

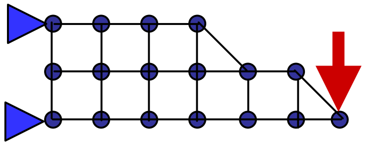
[4] Make nodes



[5] Build elements by assigning connectivity



[6] Apply boundary conditions and loads



Process and Postprocess

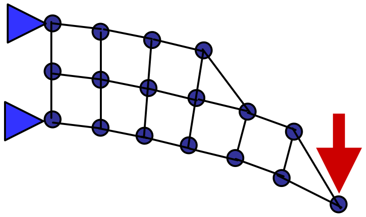
[7] Process

- Solve the boundary value problem

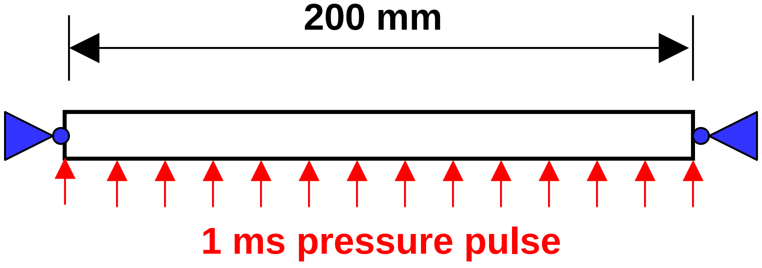
[8] Postprocess

- See the results

Displacement
Stress
Strain
Natural frequency
Temperature
Time history



Responsibility of the user



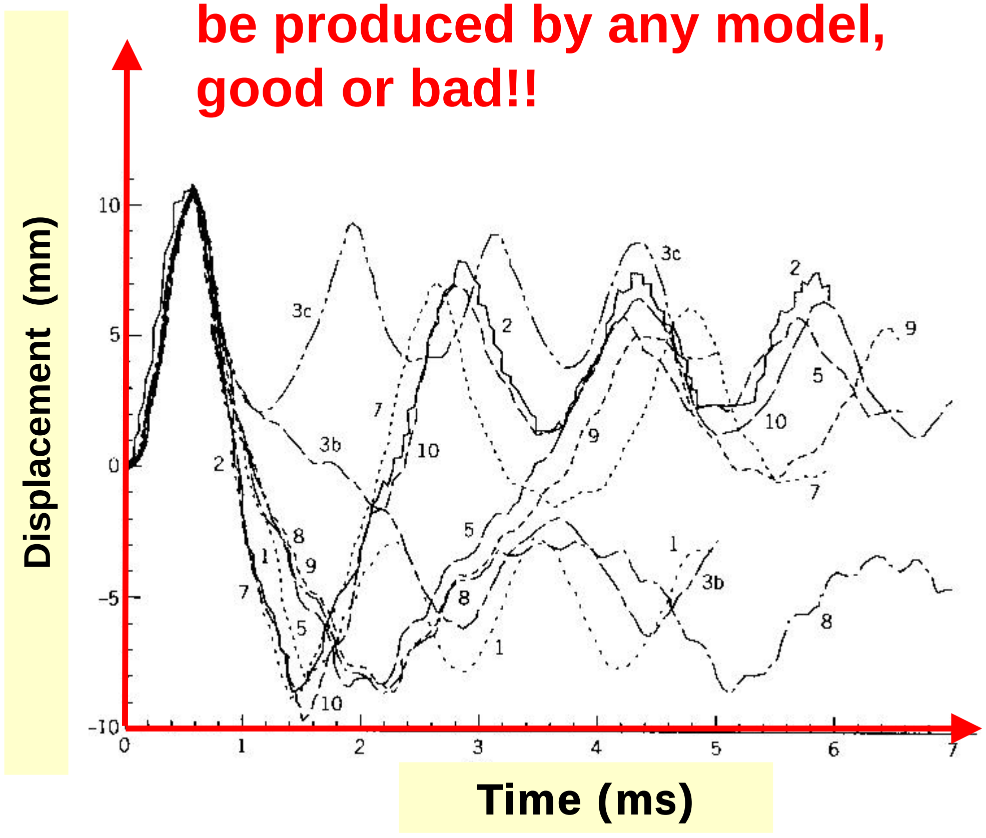
BC: Hinged supports

Load: Pressure pulse

Unknown: Lateral mid point displacement in the time domain

Results obtained from ten reputable FEM codes and by users regarded as expert.*

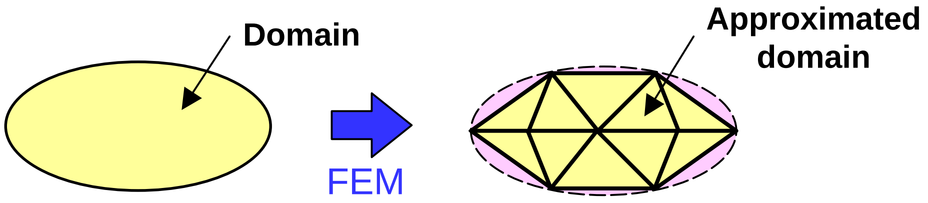
Fancy, colorful contours can be produced by any model, good or bad!!



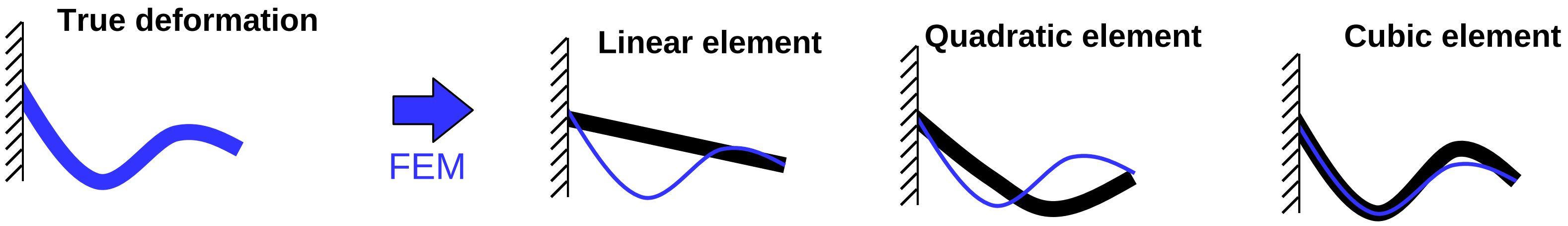
*** R. D. Cook, Finite Element Modeling for Stress Analysis, John Wiley & Sons, 1995***

Errors Inherent in FEM Formulation

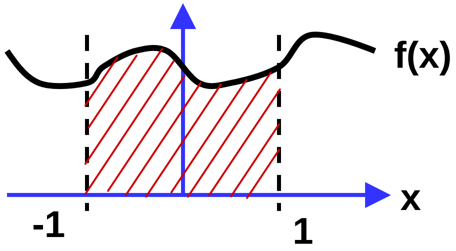
- Geometry is simplified.



- Field quantity is assumed to be a polynomial over an element. (which is not true)



- Use very simple integration techniques (Gauss Quadrature)



Area: $\int_{-1}^{1} f(x)\,dx \;\approx\; f\left(\frac{1}{\sqrt{3}}\right)+f\left(-\frac{1}{\sqrt{3}}\right)$

2-D vs. 3-D

In reality, everything is 3-D.

But some problems can be simplified to 2-D (in structures, plane stress and plane strain).

Plane Stress

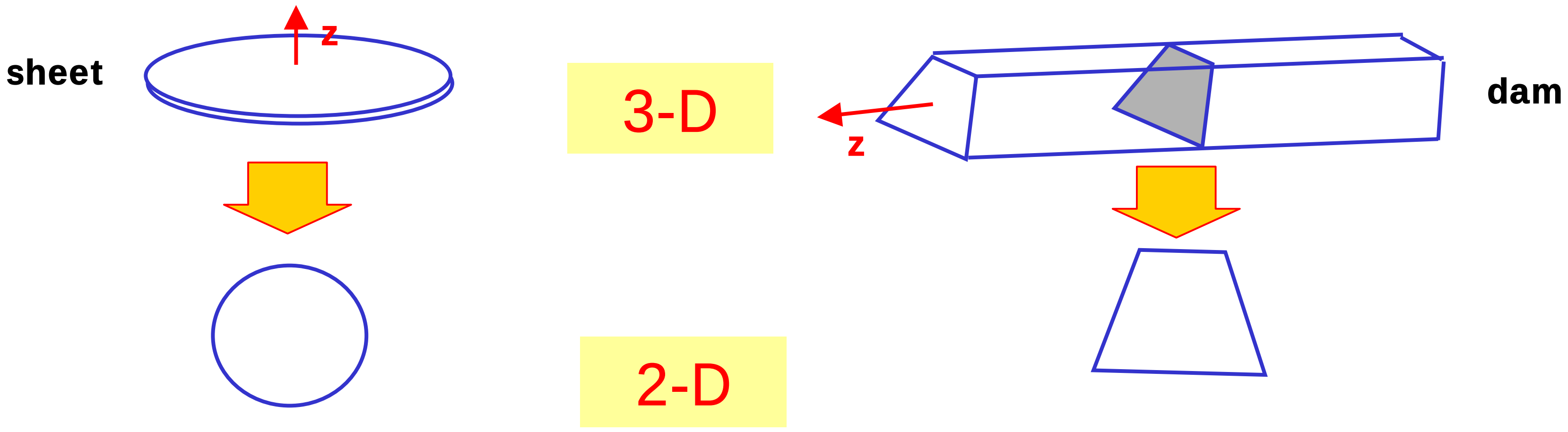
$$\sigma_z = 0$$

$$\text{thickness} \approx 0$$

Plane Strain

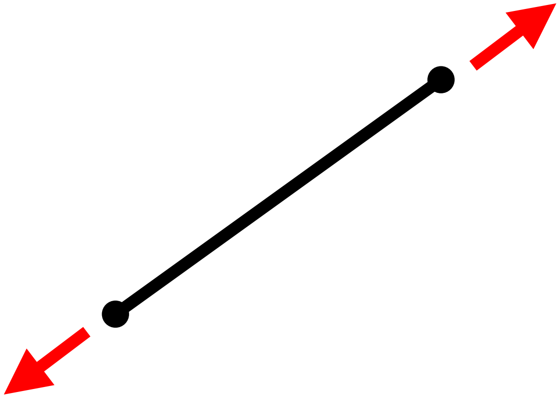
$$\varepsilon_z = 0$$

$$\text{thickness} \approx \infty$$



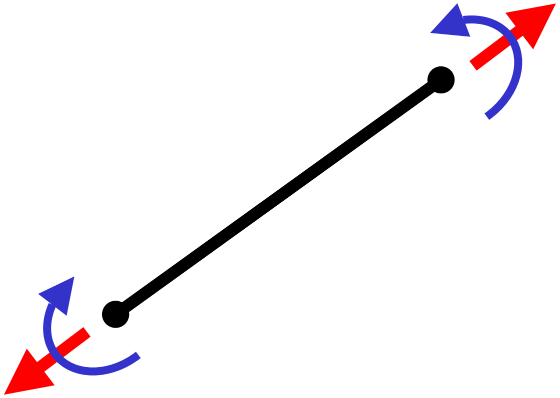
Truss vs. Beam

Truss



Only supports axial loads

Beam



Supports axial loads and bending loads

Errors Inherent in Computing

- The computer carries only a **finite number of digits**.

$$\text{e.g.)} \quad \sqrt{2} = 1.41421356, \quad \pi = 3.14159265$$

- **Numerical Difficulties**

e.g.) Very large stiffness difference

$$k_1 >> k_2 \ , \quad k_2 \approx 0$$

$$[(k_1 + k_2) - k_2]u_2 = P \ \Rightarrow u_2 = \frac{P}{k_2} \approx \frac{P}{0}$$

Mistakes by Users

- Elements are of the wrong type
  e.g) Shell elements are used where solid elements are needed

- Distorted elements

- Supports are insufficient to prevent all rigid-body motions

- Inconsistent units (e.g. E=200 GPa, Force = 100 lbs)

- Too large stiffness differences → Numerical difficulties

16.810 Plan for Today

- FEM Lecture (ca. 50 min)
  - FEM fundamental concepts, analysis procedure
  - Errors, Mistakes, and Accuracy

- **Cosmos Introduction (ca. 30 min)**
  - **Follow along step-by-step**

- **Conduct FEA of your part (ca. 90 min)**
  - **Work in teams of two**
  - **First conduct an analysis of your CAD design**
  - **You are free to make modifications to your original model**

References

Glaucio H. Paulino, *Introduction to FEM (History, Advantages and Disadvantages)*

Robert Cook et al., *Concepts and Applications of Finite Element Analysis*, John Wiley & Sons, 1989

Robert Cook, *Finite Element Modeling For Stress Analysis*, John Wiley & Sons, 1995

Introduction to Finite Element Method

J. Tinsley Oden et al., *Finite Elements – An Introduction*, Prentice Hall, 1981