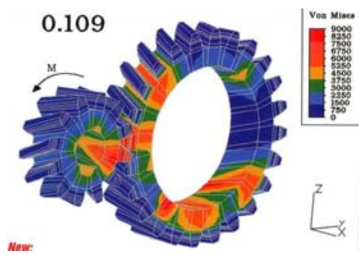
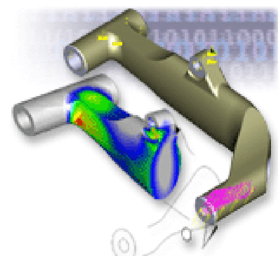
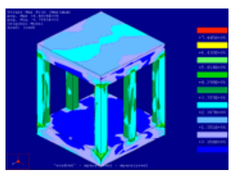
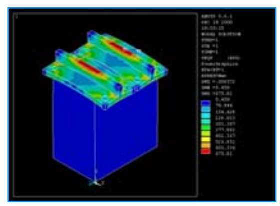
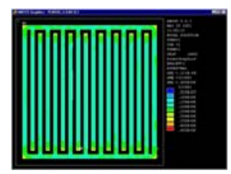
Introduction to Finite Element Analysis (FEA) or Finite Element Method (FEM)











Finite Element Analysis (FEA) or Finite Element Method (FEM)

- **The Finite Element Analysis (FEA) is a numerical method for solving problems of engineering and mathematical physics.**
- **Useful for problems with complicated geometries, loadings, and material properties where <u>analytical</u> solutions can not be obtained.**

The Purpose of FEA

In Mechanics Courses – Analytical Solution

- **Stress analysis for trusses, beams, and other simple structures are carried out based on dramatic simplification and idealization:**
  - **mass concentrated at the center of gravity**
  - **beam simplified as a line segment (same cross-section)**
- **Design is based on the calculation results of the idealized structure & a large *safety factor* (1.5-3) given by experience.**

In Engineering Design - FEA

- **Design geometry is a lot more complex; and the accuracy requirement is a lot higher. We need**
  - **To understand the physical behaviors of a complex object (strength, heat transfer capability, fluid flow, etc.)**
  - **To predict the performance and behavior of the design; to calculate the safety margin; and to identify the weakness of the design accurately; and**
  - **To identify the optimal design with confidence**

Brief History

- **Grew out of aerospace industry**
- **Post-WW II jets, missiles, space flight**
- **Need for light weight structures**
- **Required accurate stress analysis**
- **Paralleled growth of computers**

Developments

- 1940s - Hrennikoff [1941] - Lattice of 1D bars,
  - McHenry [1943] - Model 3D solids,
  - Courant [1943] - Variational form,
  - Levy [1947, 1953] - Flexibility & Stiffness
- 1950-60s - Argryis and Kelsey [1954] - Energy Principle for Matrix Methods, Turner, Clough, Martin and Topp [1956] - 2D elements, Clough [1960] - Term "Finite Elements"
- 1980s – Wide applications due to:
  - Integration of CAD/CAE – automated mesh generation and graphical display of analysis results
  - Powerful and low cost computers
- 2000s – FEA in CAD; Design Optimization in FEA; Nonlinear FEA; Better CAD/CAE Integration

FEA Applications

- **Mechanical/Aerospace/Civil/Automotive Engineering**
- **Structural/Stress Analysis**
  - **Static/Dynamic**
  - **Linear/Nonlinear**
- **Fluid Flow**
- **Heat Transfer**
- **Electromagnetic Fields**
- **Soil Mechanics**
- **Acoustics**
- **Biomechanics**

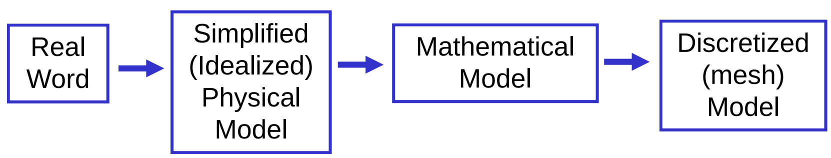
FEA

Discretization

Complex Object **Simple Analysis**

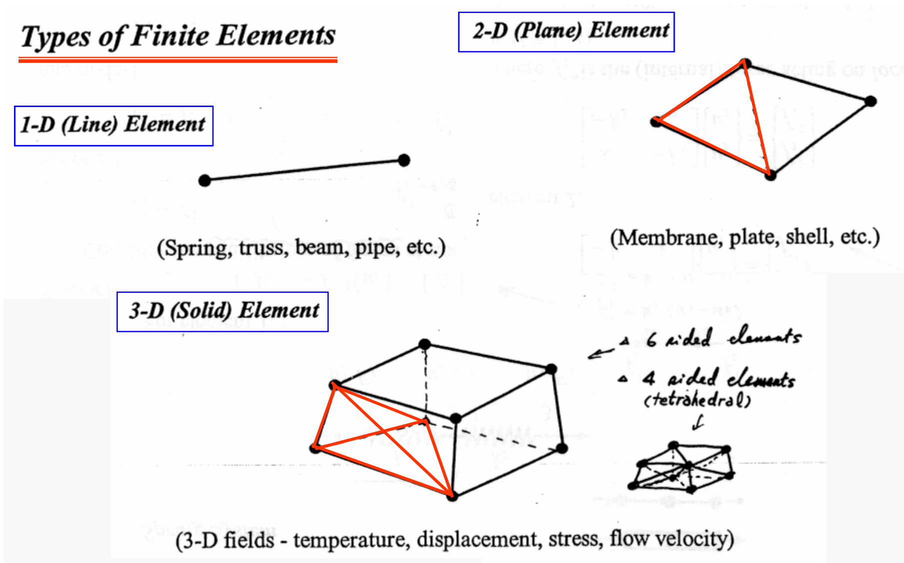
(Material discontinuity,

Complex and arbitrary geometry)



Discretizations

- **<u>Model body</u> by dividing it into <u>an equivalent system</u> of many smaller bodies or units (finite elements) interconnected at points common to two or more elements (nodes or nodal points) and/or boundary lines and/or surfaces.**



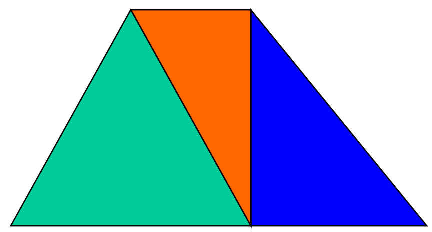
Elements & Nodes - Nodal Quantity

Feature

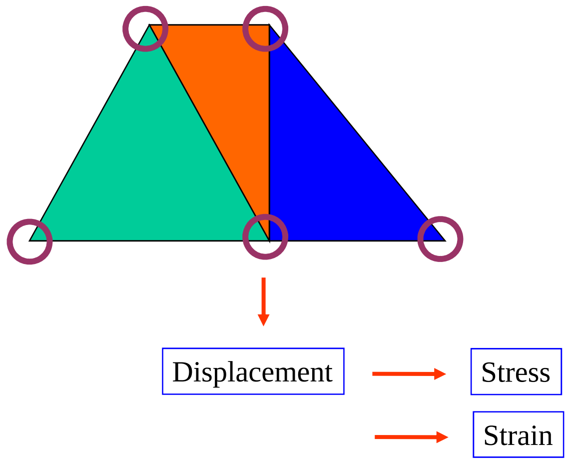
- **Obtain a set of algebraic equations to solve for unknown (first) <u>nodal quantity</u> (displacement).**
- **<u>Secondary quantities</u> (stresses and strains) are expressed in terms of nodal values of primary quantity**

Object

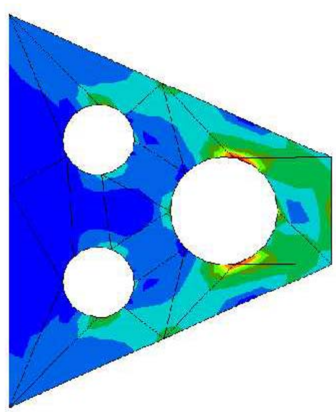
Elements



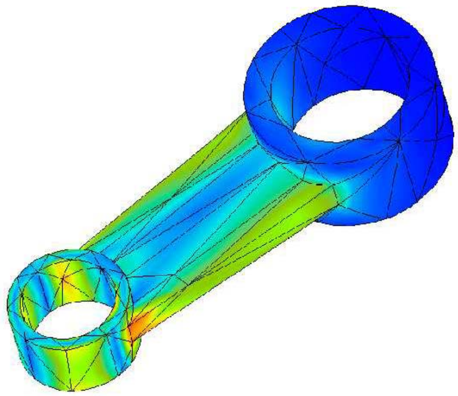
Nodes



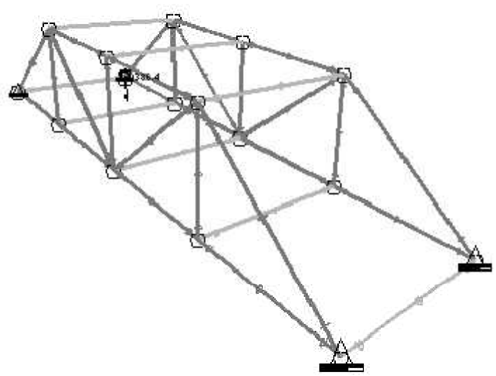
Examples of FEA - 2D



Examples of FEA – 3D



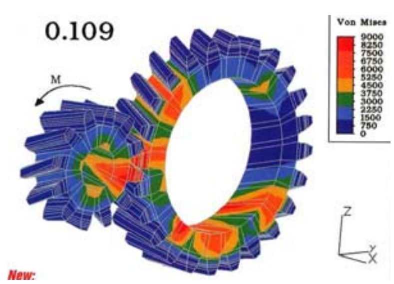
Examples of FEA

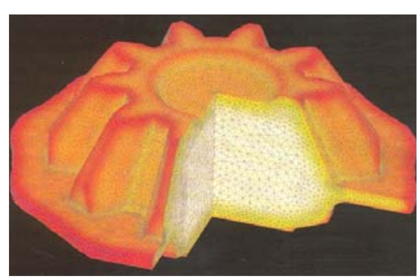


Advantages

- Irregular Boundaries
- General Loads
- Different Materials
- Boundary Conditions
- Variable Element Size
- Easy Modification
- Dynamics
- Nonlinear Problems (Geometric or Material)

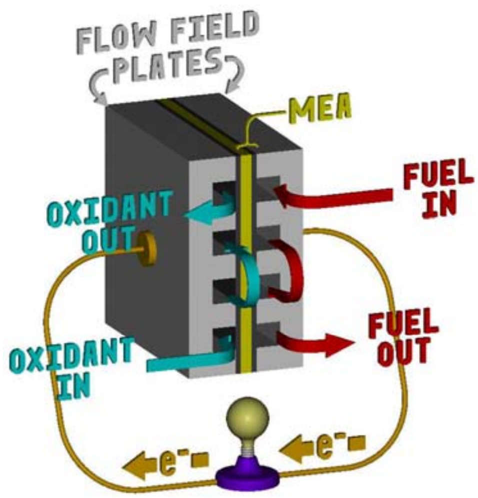
Examples for FEA





FEA in Fuel Cell Stack Design

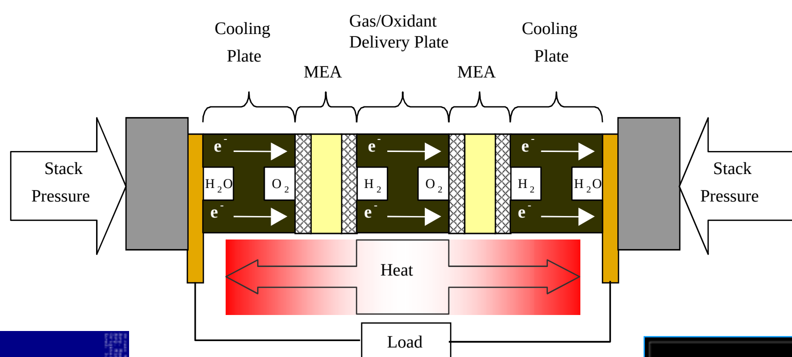
Air (O_2)

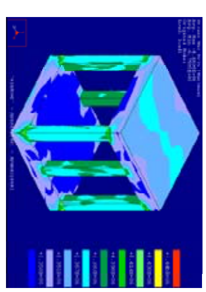


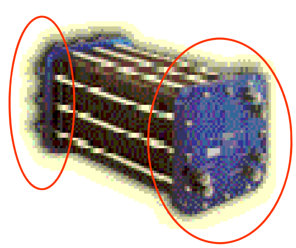
H_2

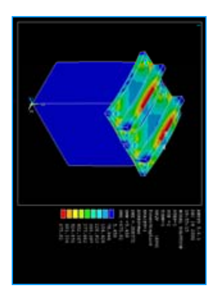
A PEM Fuel Cell

Compression of A PEM Fuel Cell Stack

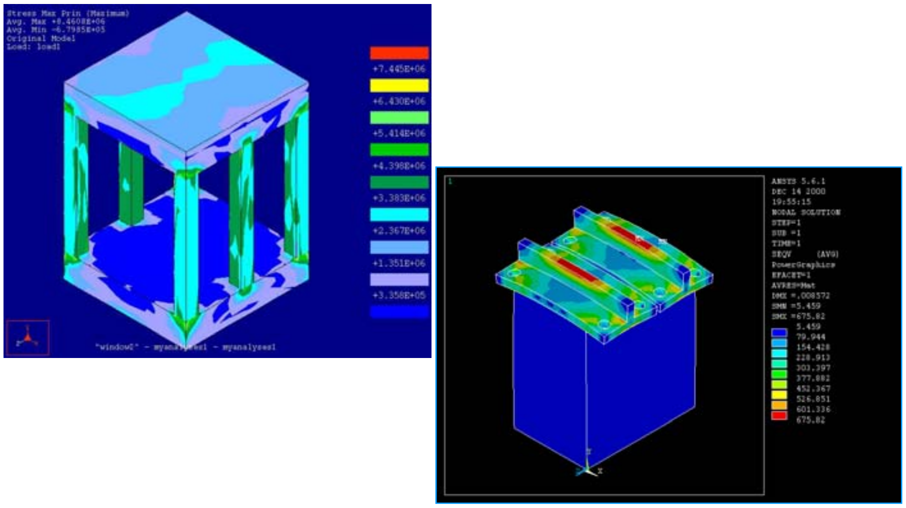




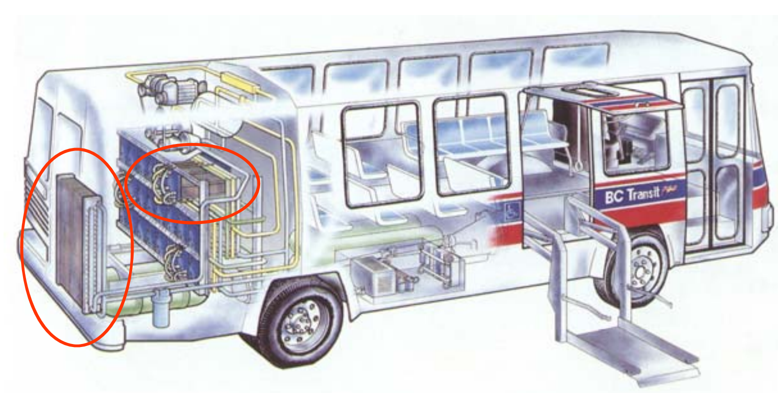




FEA on the Fuel Cell Stack and End Plate



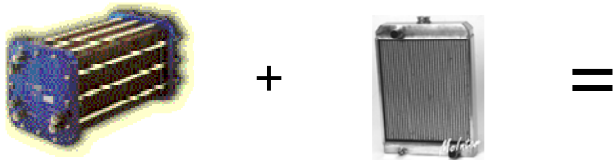
Fuel Cell in the First Ballard Prototype Bus

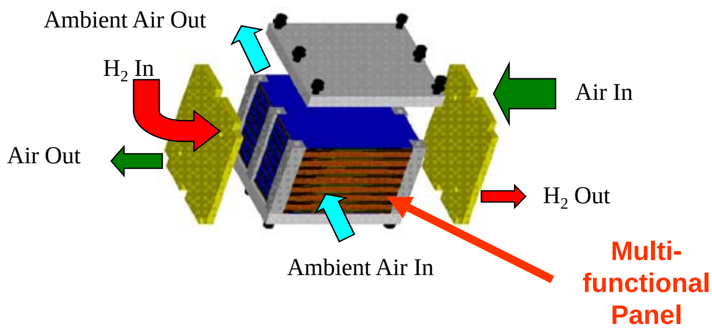


One of the key design challenge - getting rid of the low grid heat using an inefficient stainless radiator

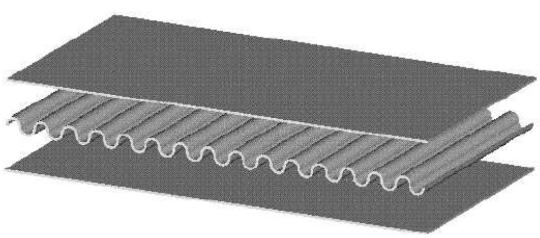
A Novel Fuel Cell Stack Design

Tri-stream, External-manifolding, Radiator Stack (TERS)





The Multi-functional Panel

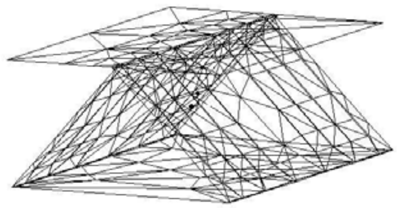


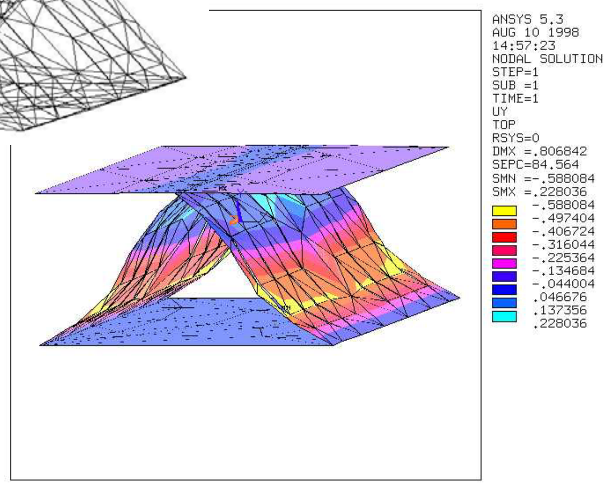
- Heat transfer and rejection
- Deformation: compensation to thermal and hydro expansion
- Electrical conductivity

Design Objective: Ideal Compression Force and Deformation - Stiffness

- cannot be achieved without modern design tool: FEA & Optimization

Stiffness Analysis and Design Optimization of the Panel





Design Variables:
- Shape
- Height
- Wavelength
- Thickness
- Surface finish
- Cuts

Principles of FEA

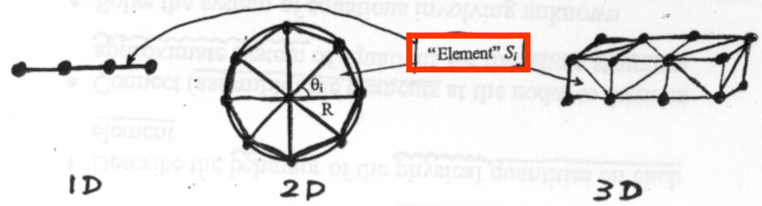
I. Basic Concepts

The *finite element method* (FEM), or *finite element analysis* (FEA), is based on the idea of building a complicated object with simple blocks, or, dividing a complicated object into small and manageable pieces. Application of this simple idea can be found everywhere in everyday life as well as in engineering.

Examples:

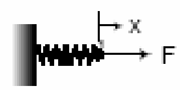
- Lego (kids' play)
- Buildings
- Approximation of the area of a circle:

Flexible, continuous body with linear deformation
↓
- finite "element"
- connecting "node"
- "mesh" of elements



Stiffness Equation

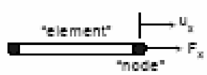
One spring



$$F = k \cdot x$$

Applied force; "stiffness" of spring (force/displacement) (lb/in); displacement of point 1

FEA w/ one DOF



$$F = k \cdot u$$

Applied force in each DOF (say x-direction); stiffness of element in each DOF (say x-direction); displacement of each DOF (say x-direction)

Note for a truss (1-D) element: $k = \dfrac{EA}{L}$

FEA for multiple (many) elements

$$\{F\} = [K] \cdot \{U\}$$

Array of applied forces (one for each DOF); Matrix of stiffnesses (DOF x DOF); Array of displacements (one for each DOF)

FEA for multiple (many) elements

$$\{F\} = [K] \cdot \{U\}$$

Array of applied forces (one for each DOF)

Matrix of stiffnesses (DOF x DOF)

Array of displacements (one for each DOF)

$\{F\}$ is "known" (loads)

$[K]$ is "known" (geometry, material properties…elements)

$\{U\}$ is to be determined (displacements)

This can be solved mathematically using a matrix inversion method

$$\{F\} = [K] \cdot \{U\} \quad \rightarrow \quad \{U\} = [K]^{-1}\{F\}$$

(first nodal quantity)

Once the displacements $\{U\}$ are known, then strains and stresses can be determined:

$\varepsilon = \dfrac{\Delta u}{L}$ (1-D …more complicated for 2-D and 3-D strains)

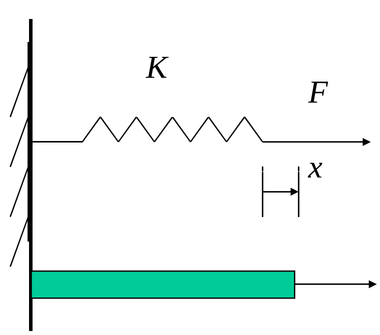
$\sigma = E \cdot \varepsilon$

and $FOS = \dfrac{\sigma_y}{\sigma}$

(second nodal quantities)

A Simple Stiffness Equation

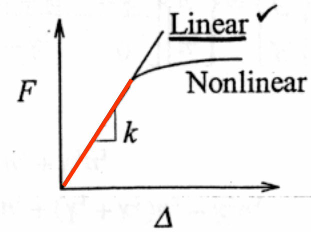
$$Kx = F$$



Simplest

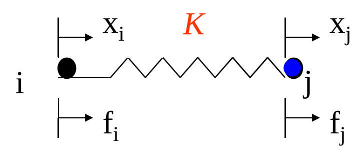
Spring force-displacement relationship:

$F = k\Delta$ with $\Delta = u_j - u_i$



$k = F/\Delta$ (> 0) is the force needed to produce a unit stretch.

Stiffness Equation of One Spring

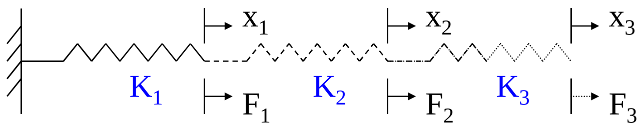


$$\begin{cases} K(x_i - x_j) = f_i \\ -K(x_i - x_j) = f_j \end{cases} \rightarrow \begin{bmatrix} k_{ii} & k_{ij} \\ k_{ji} & k_{jj} \end{bmatrix} \begin{bmatrix} x_i \\ x_j \end{bmatrix} = \begin{bmatrix} f_i \\ f_j \end{bmatrix}$$

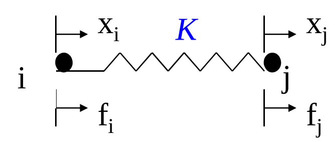
By using the unit displacement method, we can express the stiffness coefficients k_{ij} etc. in terms of the spring coefficient K

$$k_{ii} = k_{jj} = K, \quad k_{ij} = k_{ji} = -K$$

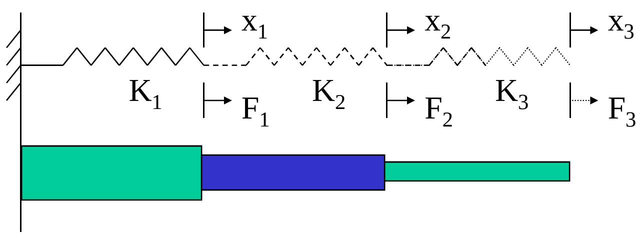
The Three-Spring System



- **No geometry influence**
- **Simple material property *(K)***
- **Simple load condition**
- **Simple constraints (boundary condition)**



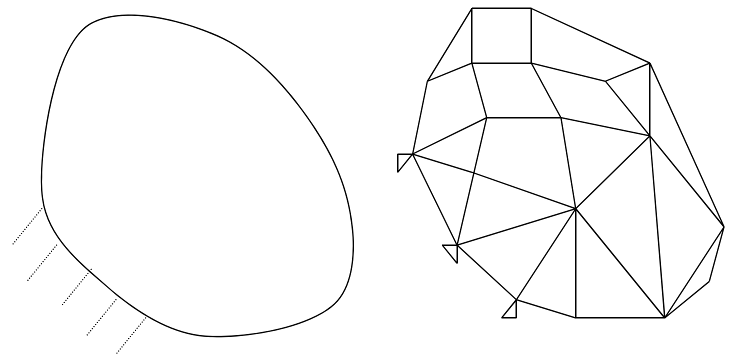
Identify and Solve the Stiffness Equations for a System of "Finite Elements"



$$\begin{bmatrix} x_1 \\ x_2 \\ x_3 \end{bmatrix} = \begin{bmatrix} F_1 \\ F_2 \\ F_3 \end{bmatrix}$$

Medium

An Elastic Solid --> A System of "Springs"



Task of FEA: To identify and solve the stiffness equations for a system of "finite elements."

Real-Complex

An Elastic Solid → A System of "Springs"

- The actual solid (plate, shell, etc.) is discretized into a number of smaller units called elements.
- These small units have finite dimensions - hence the word ***finite element***.
- The discrete "**equivalent spring**" system provides an approximate model for the actual elastic body
- It is reasonable to say that the larger the number of elements used, the better will be the approximation
- Think "spring" as one type of elastic units; we can use other types such as truss, beam, shell, etc.

Real-Complex

A System of Springs under A Number of Forces

- The system's configuration will change.
- We need to measure deflections at several points to characterize such changes.
- A system of linear equations is introduced.

$$\begin{bmatrix} k_{11} & k_{12} & \cdots & k_{1n} \\ k_{21} & k_{22} & \cdots & k_{2n} \\ \vdots & \vdots & \vdots & \vdots \\ k_{n1} & k_{n2} & \cdots & k_{nn} \end{bmatrix} \begin{bmatrix} x_1 \\ x_2 \\ \vdots \\ x_n \end{bmatrix} = \begin{bmatrix} F_1 \\ F_2 \\ \vdots \\ F_n \end{bmatrix}$$

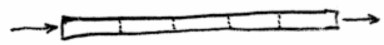
Real-Complex

Procedure for Carrying out Finite Element Analysis

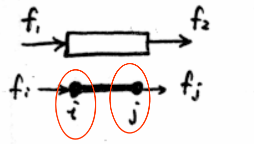
To construct the stiffness equations of a complex system made up of springs, one need to develop the **stiffness equation** of ***one spring*** and use the equation as a ***building block***

- Stiffness equation of one spring/block
- Way of stacking blocks

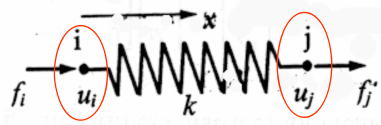
Spring Element



"*Everything important is simple.*"

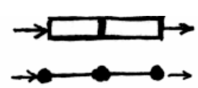


One Spring Element

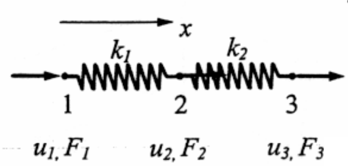


Two nodes:	i, j
Nodal displacements:	u_i, u_j (in, m, mm)
Nodal forces:	f_i, f_j (lb, Newton)
Spring constant (stiffness):	k (lb/in, N/m, N/mm)

Spring System

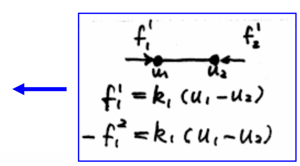


3 nodes
2 elements



For element 1,

$$\begin{bmatrix} k_1 & -k_1 \\ -k_1 & k_1 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} f_1^1 \\ f_2^1 \end{Bmatrix}$$

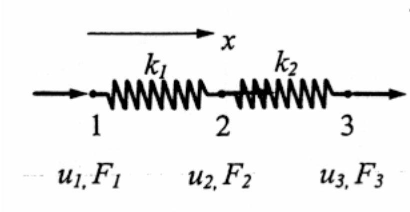


element 2,

$$\begin{bmatrix} k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} f_1^2 \\ f_2^2 \end{Bmatrix}$$

where f_i^m is the (internal) force acting on *local* node *i* of element *m* (*i* = 1, 2).

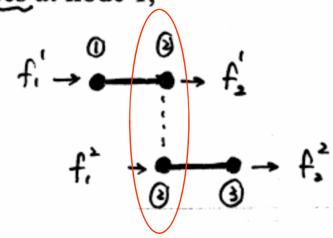
3 nodes
2 elements



Assemble the stiffness matrix for the whole system:

Consider the equilibrium of forces at node 1,

$$F_1 = f_1^1$$



at node 2,

$$F_2 = f_2^1 + f_1^2$$

and node 3,

$$F_3 = f_2^2$$

$$F_1 = k_1 u_1 - k_1 u_2$$
$$F_2 = -k_1 u_1 + (k_1 + k_2) u_2 - k_2 u_3$$
$$F_3 = -k_2 u_2 + k_2 u_3$$

$K \times \Delta = F$

In matrix form,

$$\begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1 + k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix}$$

(assembly)

or

$$\mathbf{KU} = \mathbf{F}$$

K is the stiffness matrix (structure matrix) for the spring system.

An alternative way of assembling the whole stiffness matrix:

"Enlarging" the stiffness matrices for elements 1 and 2, we have

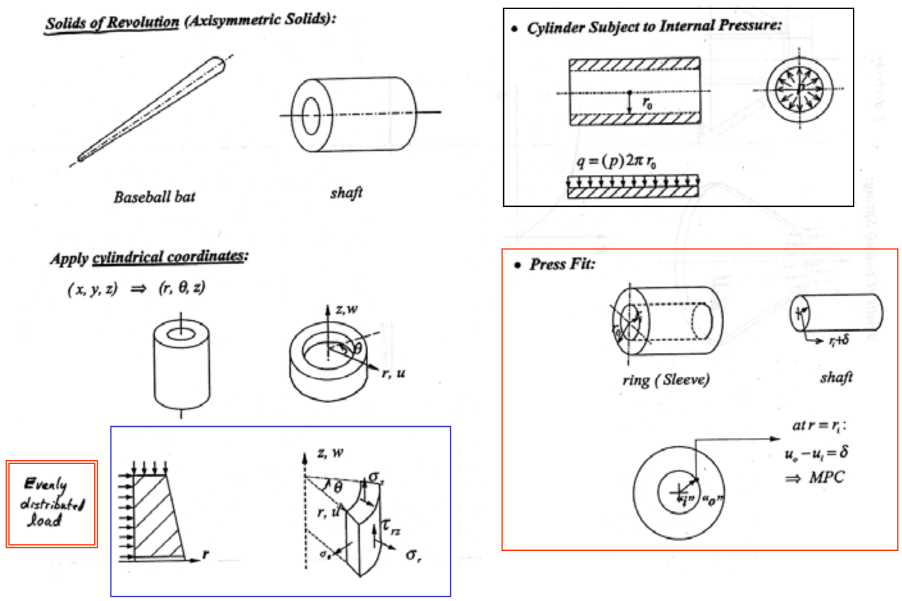
$$\begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1 & 0 \\ 0 & 0 & 0 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} f_1^1 \\ f_2^1 \\ 0 \end{Bmatrix}$$

$$\begin{bmatrix} 0 & 0 & 0 \\ 0 & k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ f_1^2 \\ f_2^2 \end{Bmatrix}$$

Way of Stacking Blocks/Elements

- **Compatibility requirement: ensures that the "displacements" at the shared node of adjacent elements are equal.**
- **Equilibrium requirement: ensures that elemental forces and the external forces applied to the system nodes are in equilibrium.**
- **Boundary conditions: ensures the system satisfy the boundary constraints and so on.**

Applying Different Types of Loads



FEM in Structural Analysis

Procedures:

- Divide structure into pieces (elements with nodes)
- Describe the behavior of the physical quantities on each element
- Connect (assemble) the elements at the nodes to form an approximate system of equations for the whole structure
- Solve the system of equations involving unknown quantities at the nodes (e.g., displacements)
- Calculate desired quantities (e.g., strains and stresses) at selected elements
- **Interpret the Results**

Integrated CAD/CAE System – Automated FEA

Computer Implementations

- Preprocessing (build FE model, loads and constraints)
- FEA solver (assemble and solve the system of equations)
- Postprocessing (sort and display the results)

from CAD Model / FEM User Interface; ① simplified geometry, ②, ③; graphically using colors

Detailed Process – Pro/MECHANICA

Commercial FEA Software

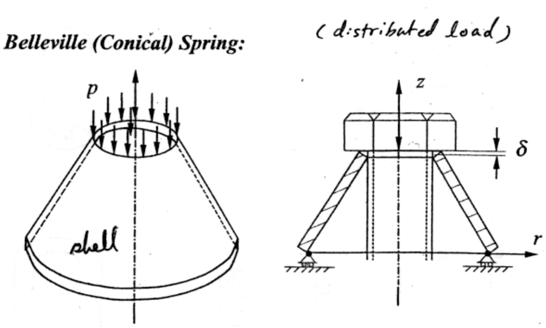
- Pro/MECHANICA
- ANSYS
- ALGOR
- COSMOS
- STARDYNE
- IMAGES-3D
- MSC/NASTRAN
- SAP90
- SDRC/I-DEAS
- ADINA
- NISA
- …

Advantages of General Purpose Programs

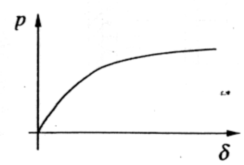
- Easy input - preprocessor
- Solves many types of problems
- Modular design - fluids, dynamics, heat, etc.
- Can run on PC's now.
- Relatively low cost.

Limitations of Regular FEA Software

- Belleville (Conical) Spring:



- Unable to handle geometrically nonlinear - large deformation problems: shells, rubber, etc.



This is a geometrically nonlinear (large deformation) problem and iteration method (incremental approach) needs to be employed.