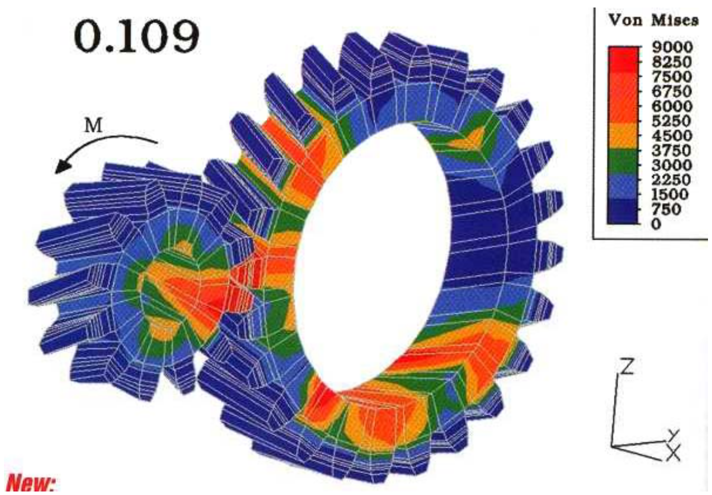
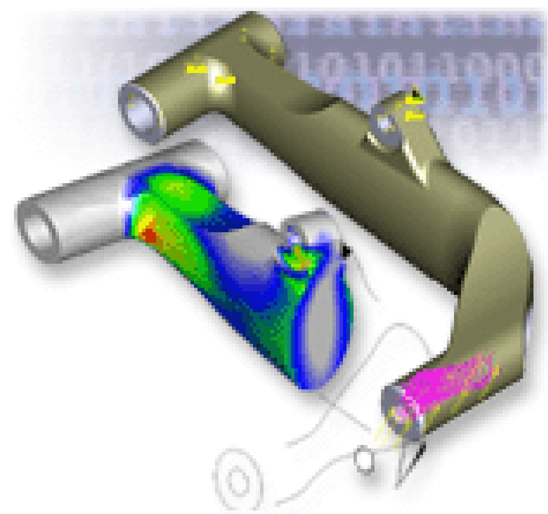
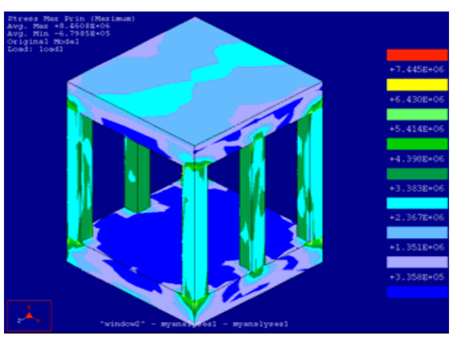
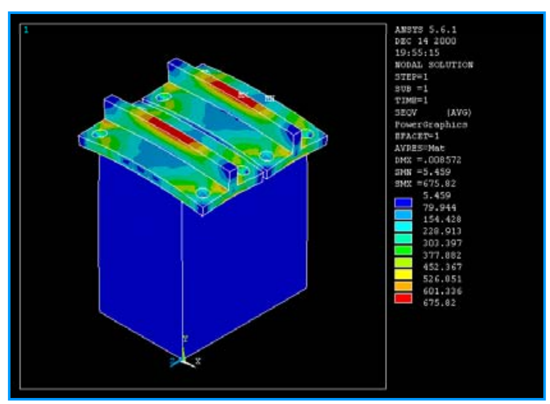
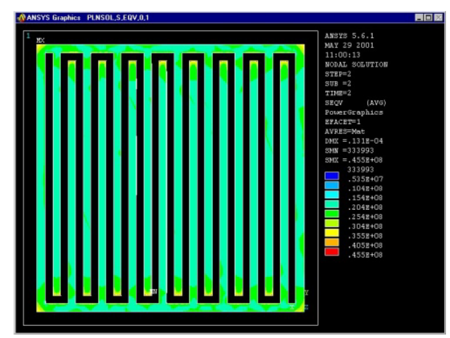
Introduction to Finite Element Analysis (FEA) or Finite Element Method (FEM)











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 analytical 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 <u>dramatic simplification and idealization:</u>**
  - **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 <u>idealized</u> structure & a large <u>*safety factor* (1.5-3)</u> 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 <u>complex</u> 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 <u>accurately</u>; and**
  - **To identify the optimal design with <u>confidence</u>**

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)

Real Word → Simplified (Idealized) Physical Model → Mathematical Model → Discretized (mesh) Model

Discretizations

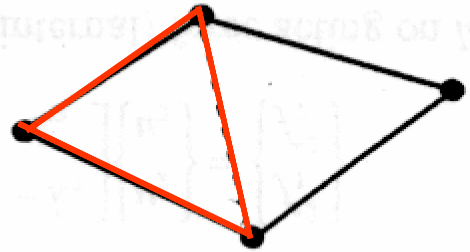
- Model body by dividing it into an equivalent system of many smaller bodies or units (finite elements) interconnected at points common to two or more elements (nodes or nodal points) and/or boundary lines and/or surfaces.

Types of Finite Elements

1-D (Line) Element

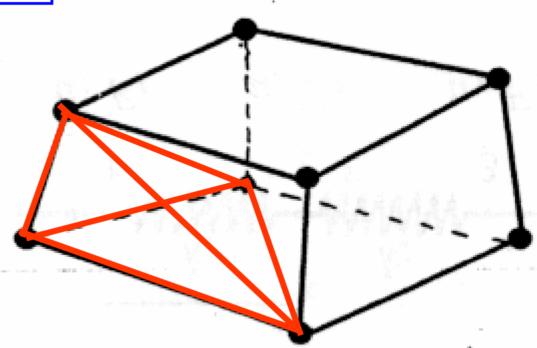
(Spring, truss, beam, pipe, etc.)

2-D (Plane) Element

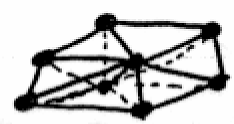


(Membrane, plate, shell, etc.)

3-D (Solid) Element



- 6 sided elements
- 4 sided elements (tetrahedral)



(3-D fields - temperature, displacement, stress, flow velocity)

Elements & Nodes - Nodal Quantity

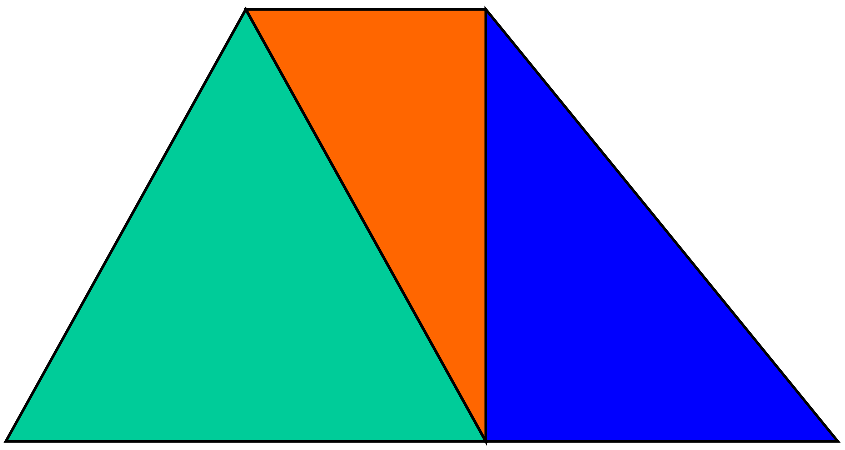
Feature

- **Obtain a set of algebraic equations to solve for unknown (first) <u>nodal quantity</u> (displacement).**

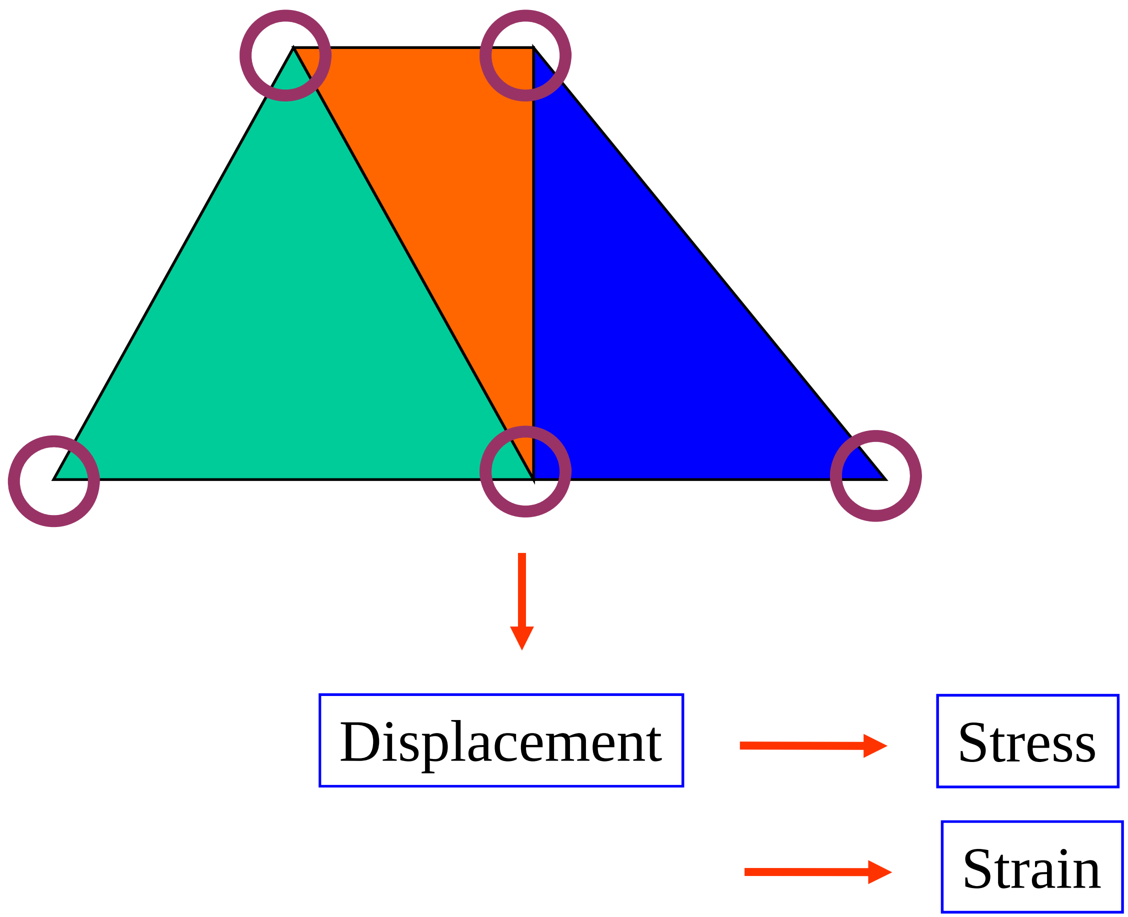
- **<u>Secondary quantities</u> (stresses and strains) are expressed in terms of nodal values of primary quantity**

Object

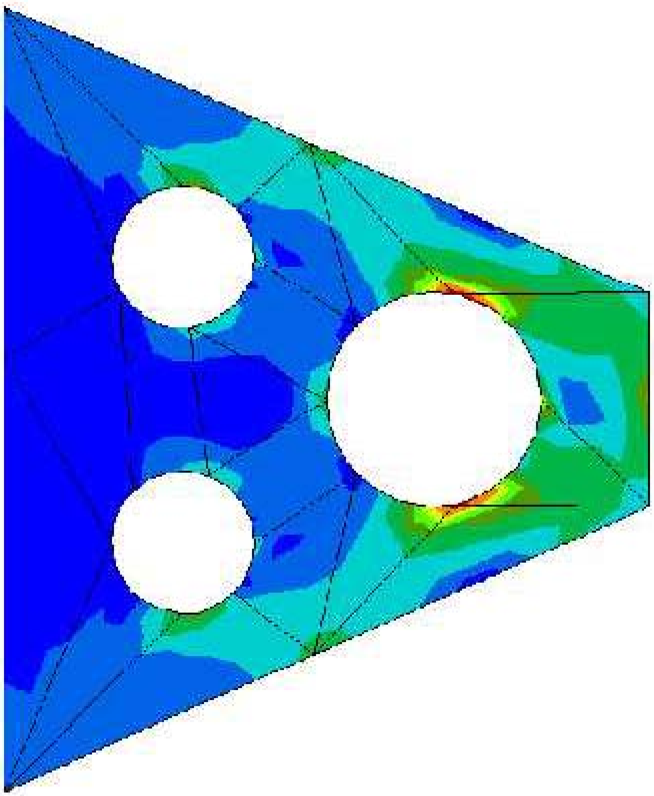
Elements



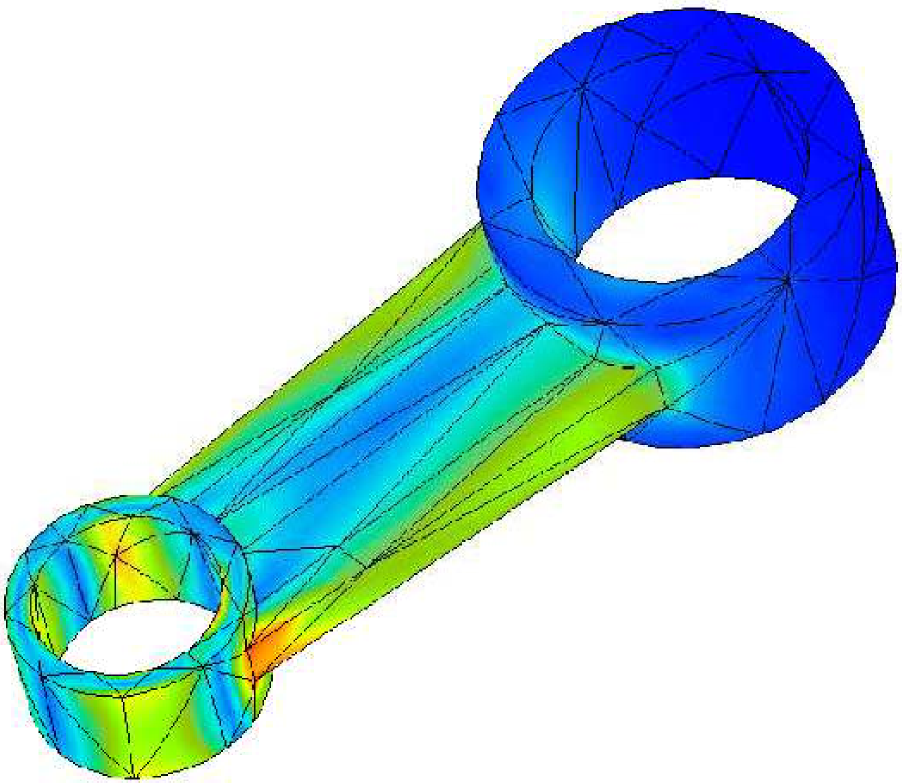
Nodes



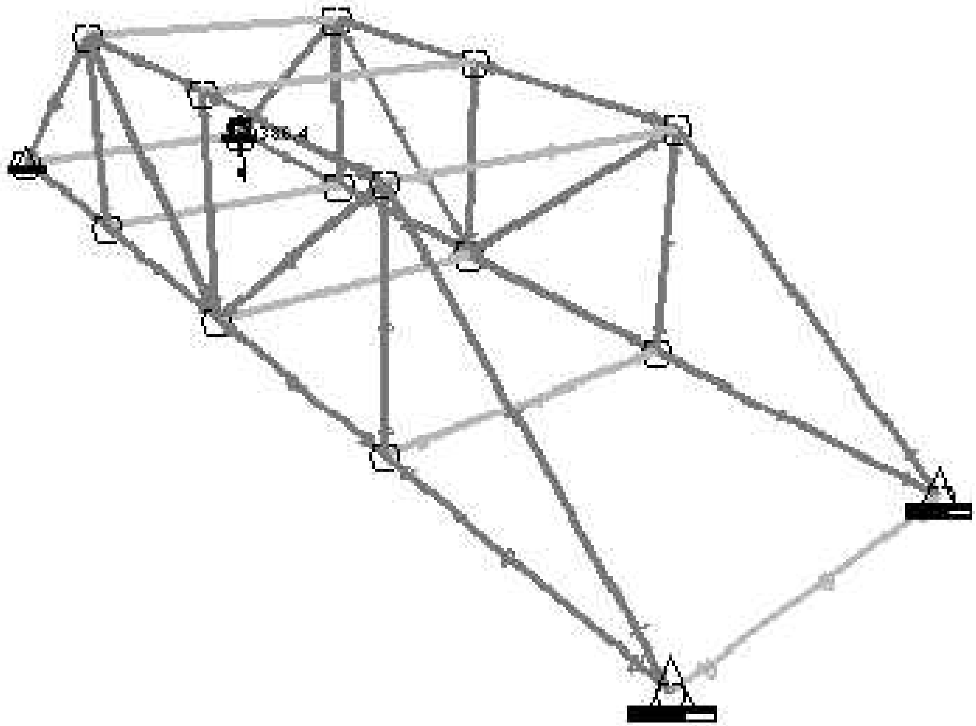
Examples of FEA - 2D



Examples of FEA – 3D



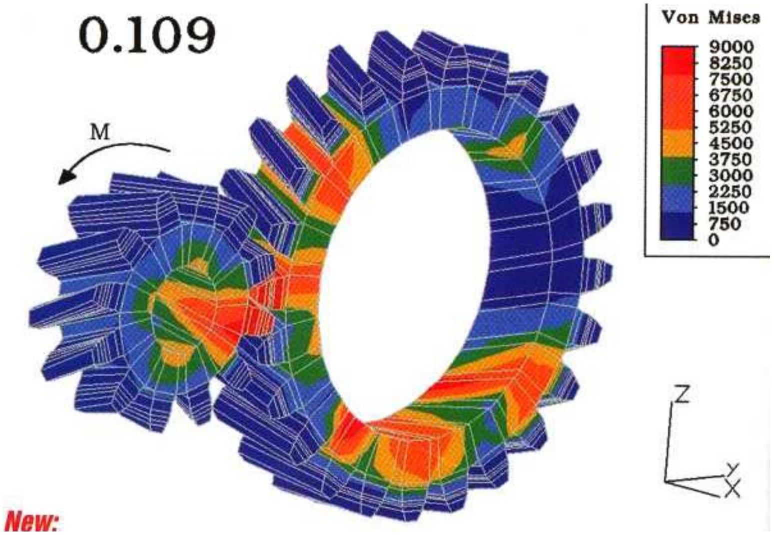
Examples of FEA

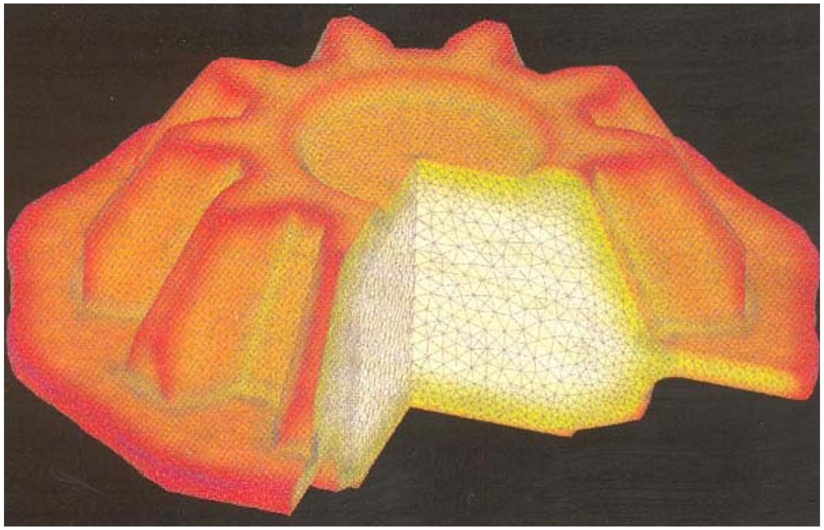


Advantages

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

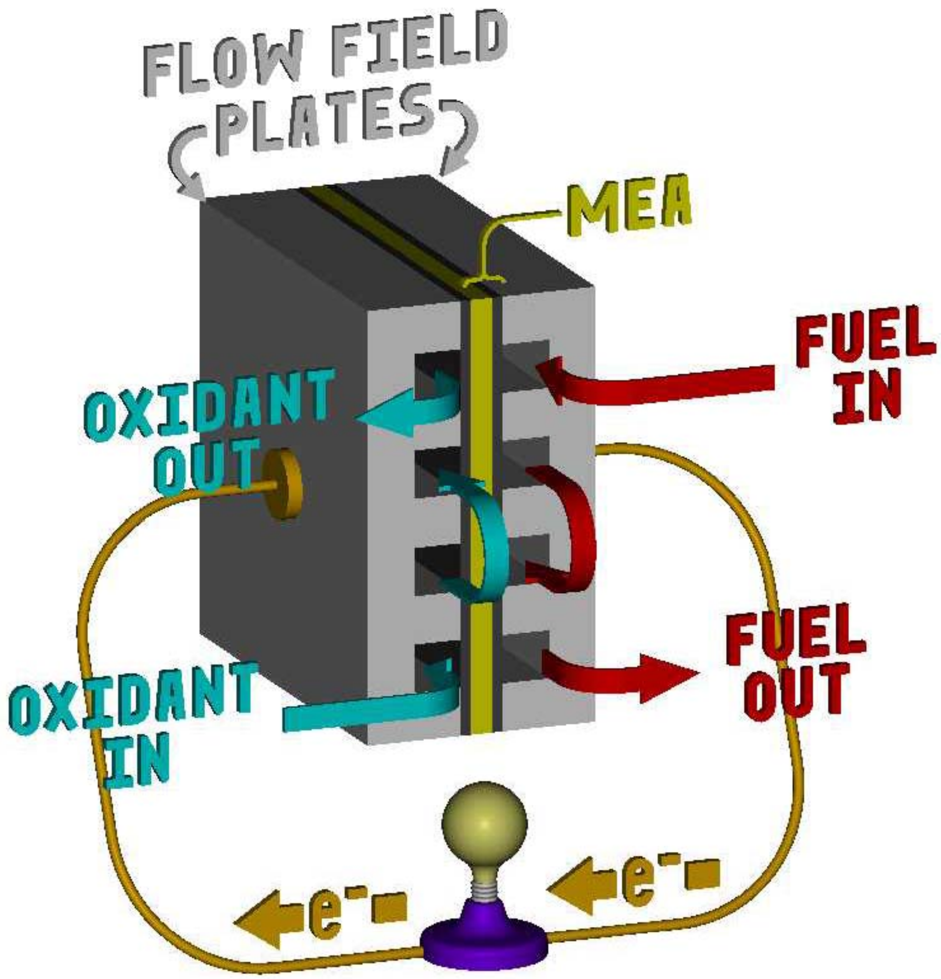
Examples for FEA





FEA in Fuel Cell Stack Design

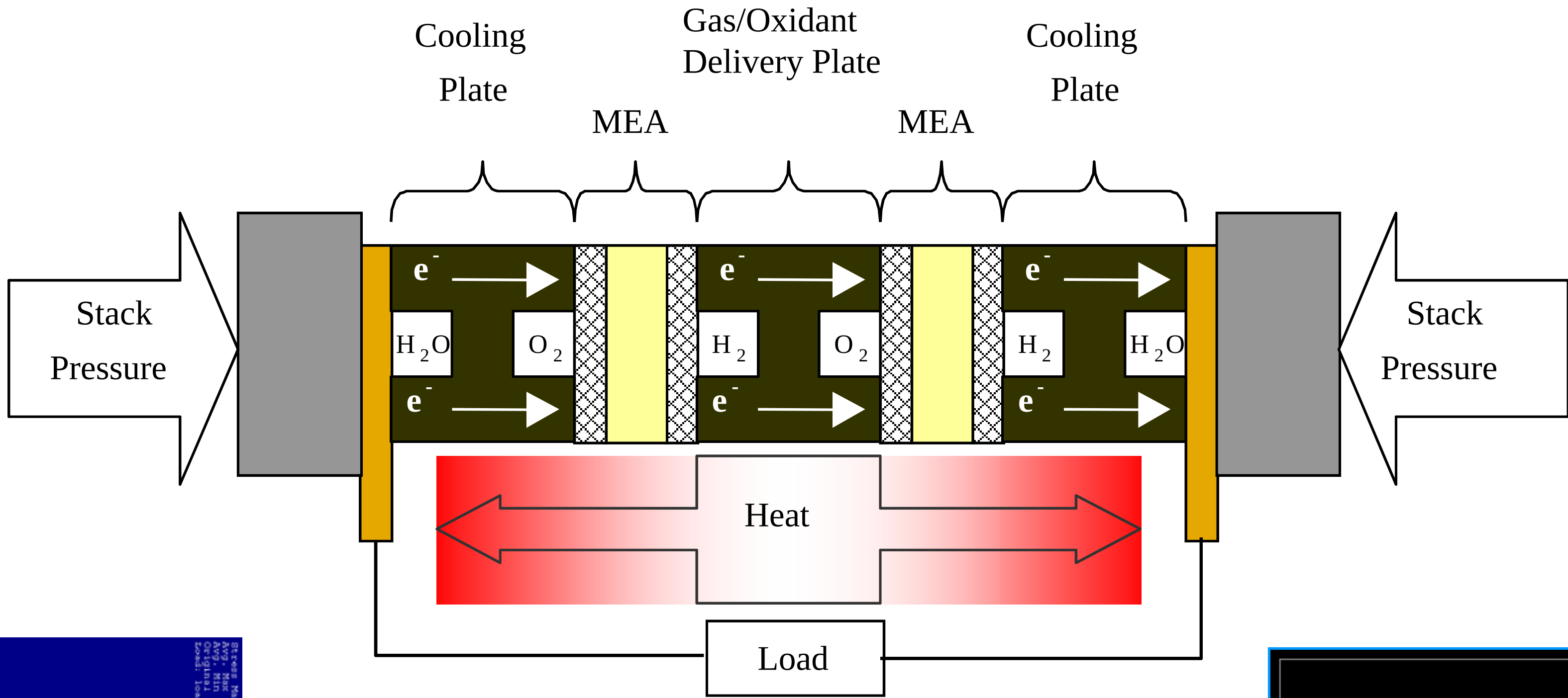
Air (O_2)

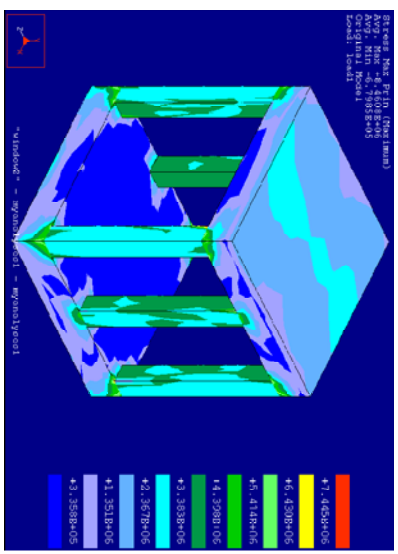


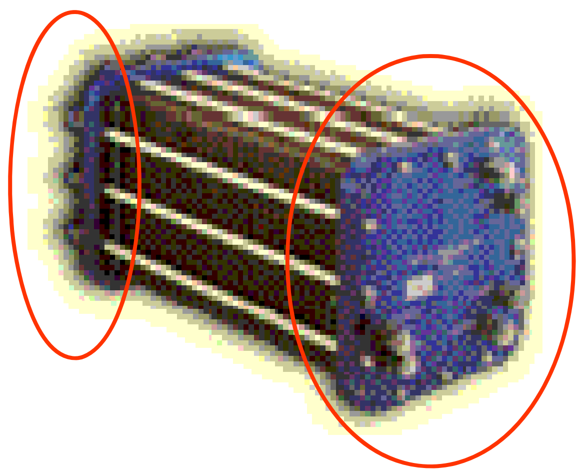
H_2

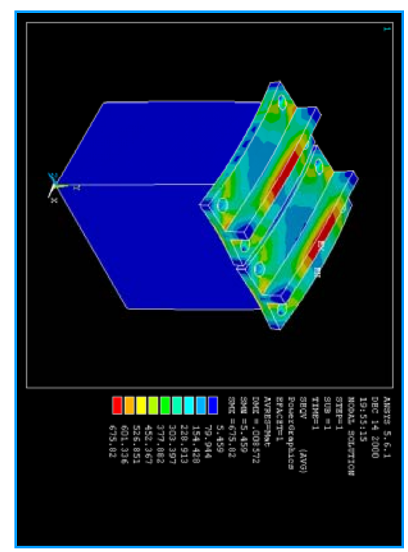
A PEM Fuel Cell

Compression of A PEM Fuel Cell Stack

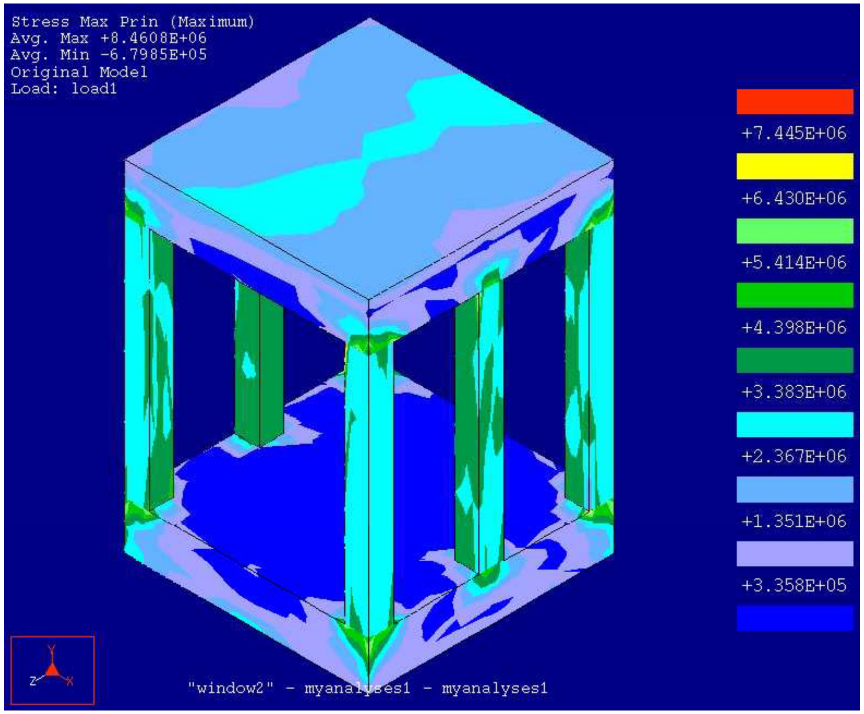


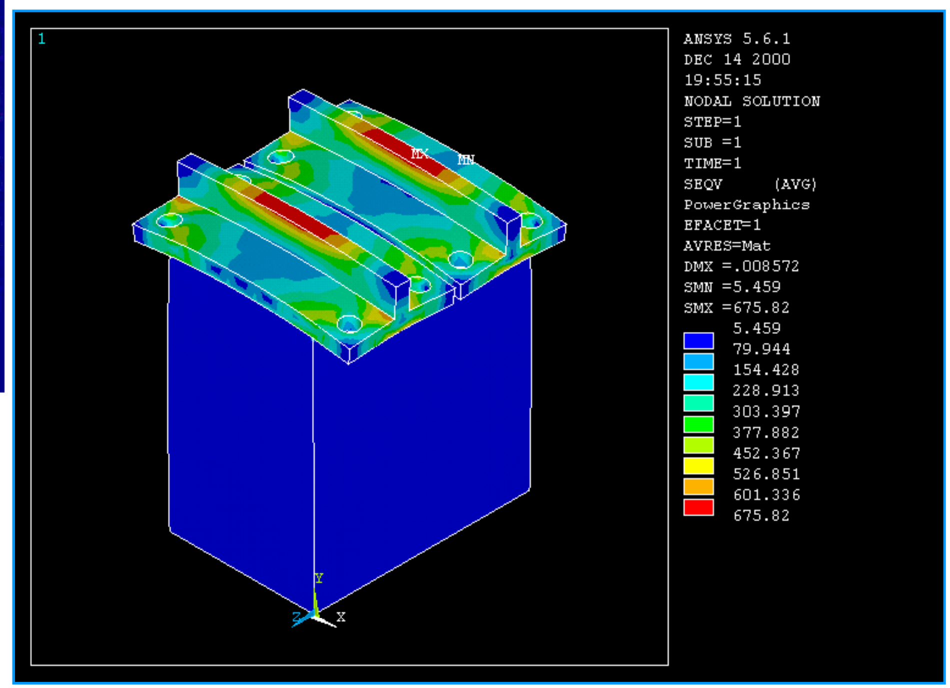




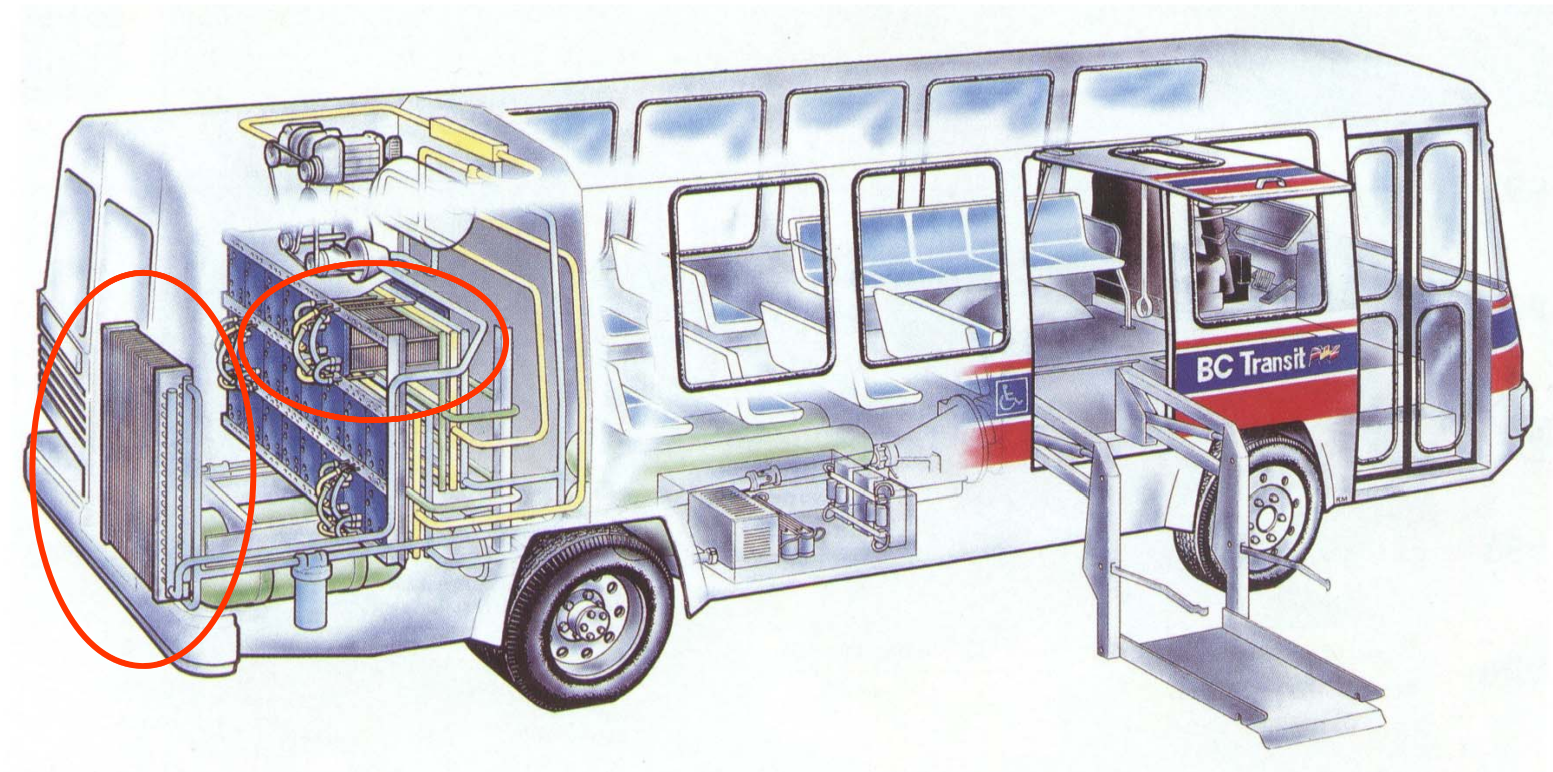


FEA on the Fuel Cell Stack and End Plate





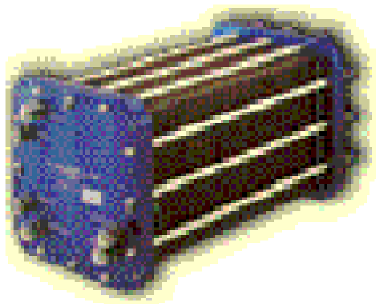
Fuel Cell in the First Ballard Prototype Bus



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

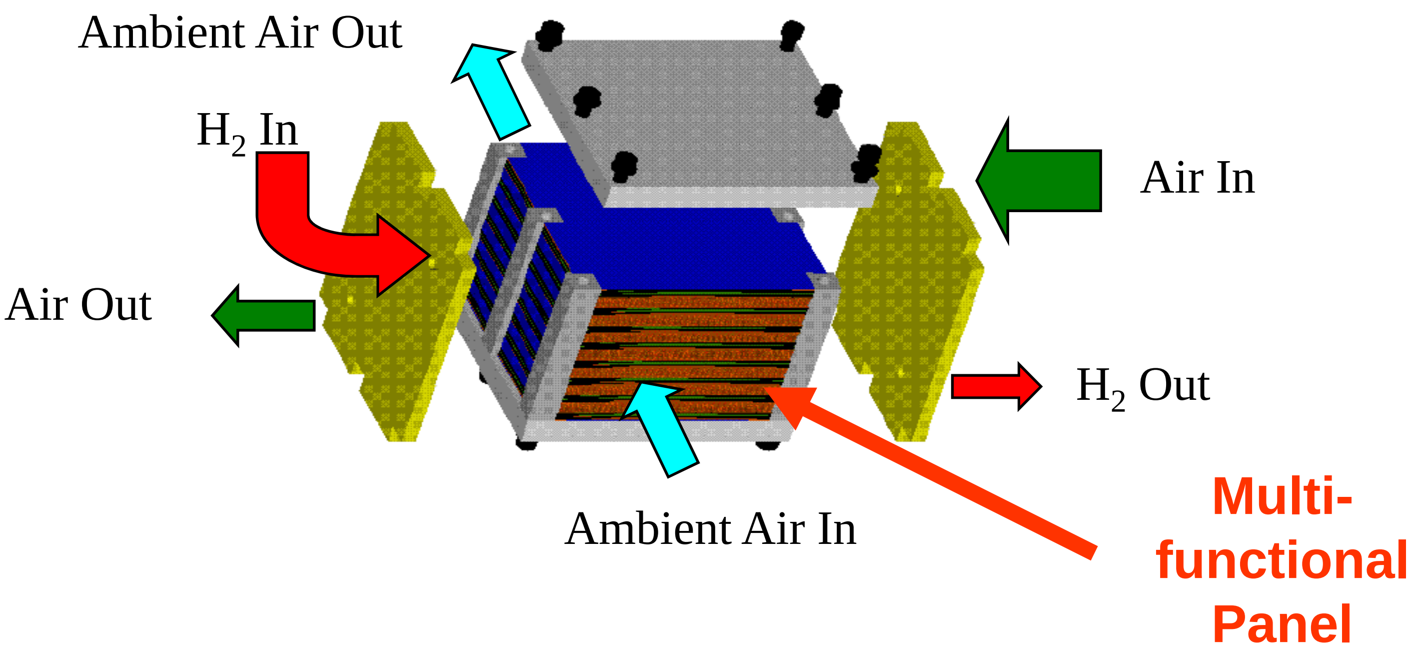
A Novel Fuel Cell Stack Design

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

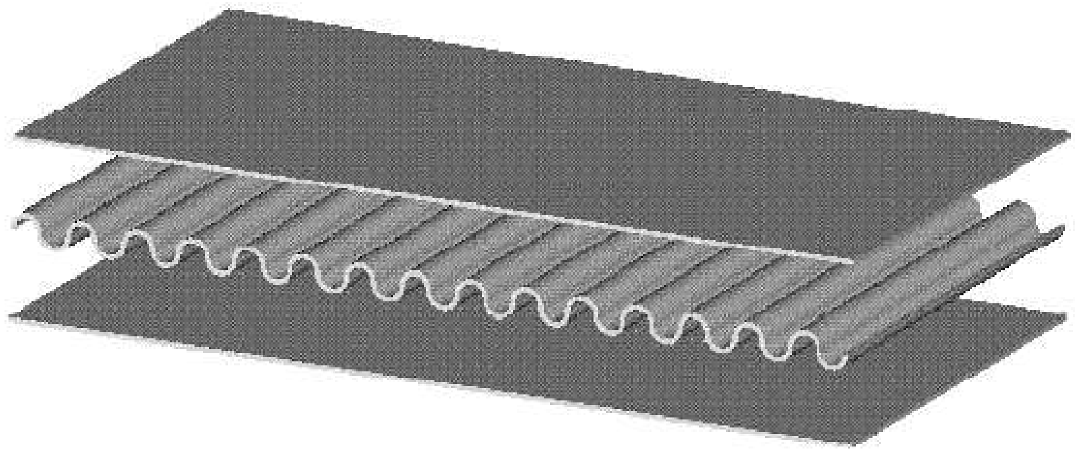


+

=



The Multi-functional Panel

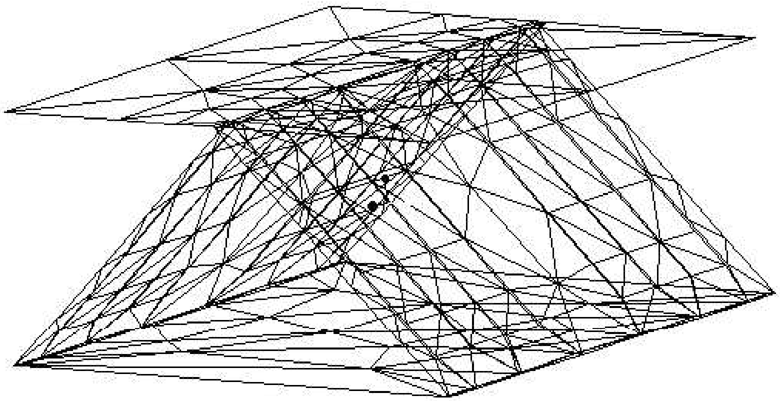


- Heat transfer and rejection
- Deformation: compensation to thermal and hydro expansion
- Electrical conductivity

Design Objective: Ideal Compression Force and Deformation - Stiffness

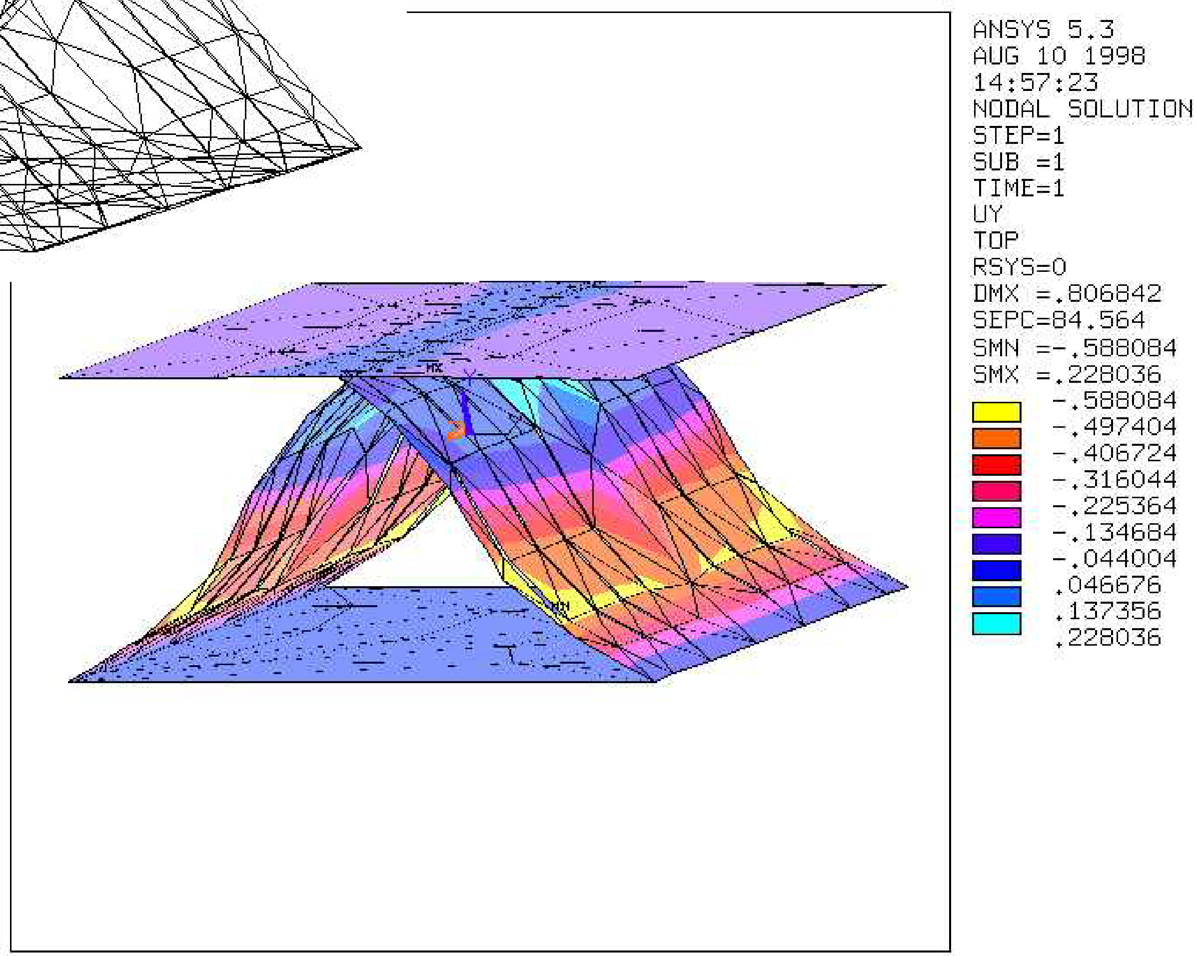
- cannot be achieved without modern design tool: FEA & Optimization

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.

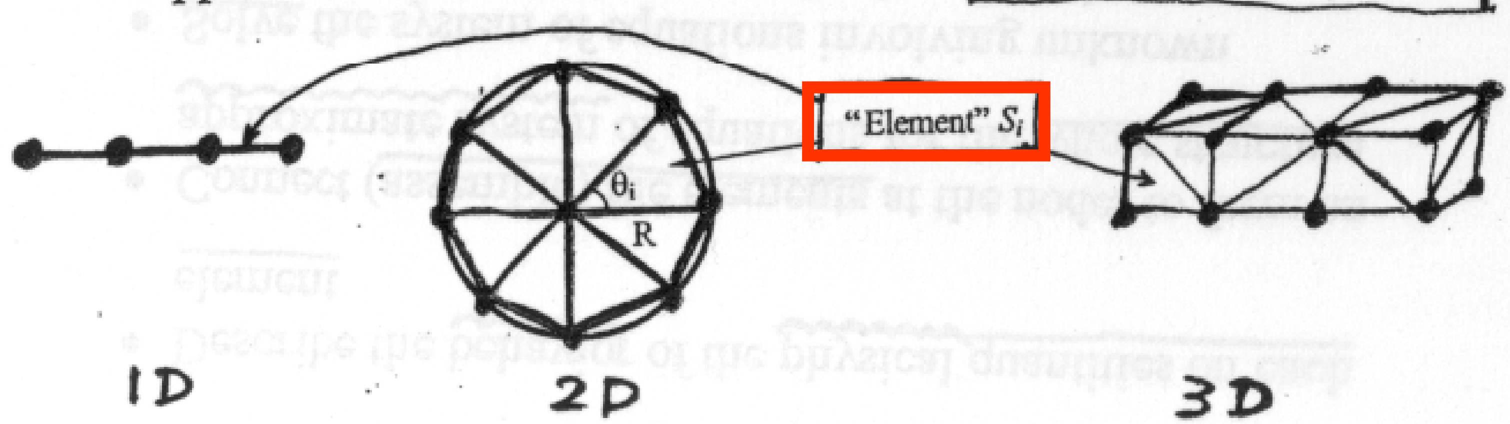
Examples:

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

Flexible, continuous body with linear deformation

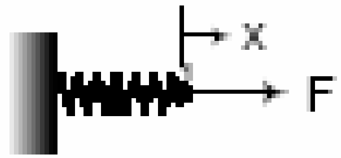
↓

- finite "element"
- connecting "node"
- "mesh" of elements



Stiffness Equation

One spring



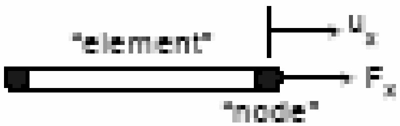
$$F = k \cdot x$$

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:

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

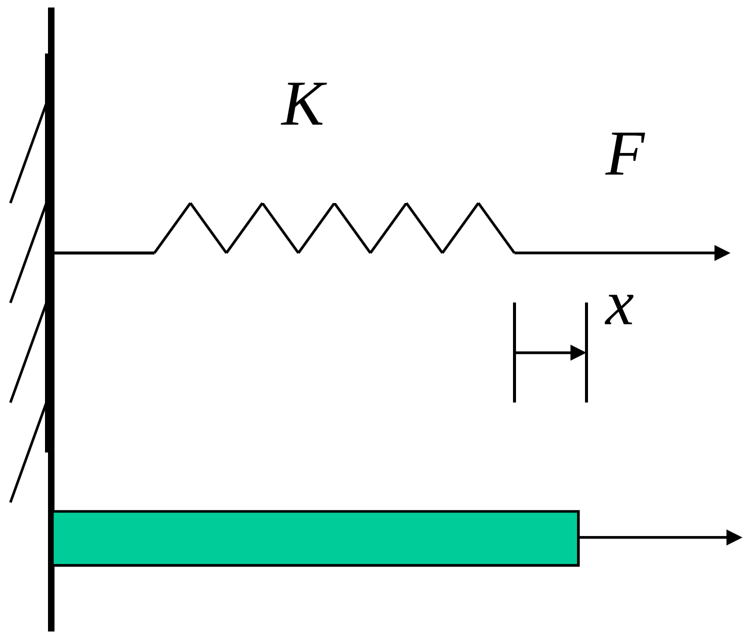
$$\sigma = E \cdot \varepsilon$$

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

(second nodal quantities)

A Simple Stiffness Equation

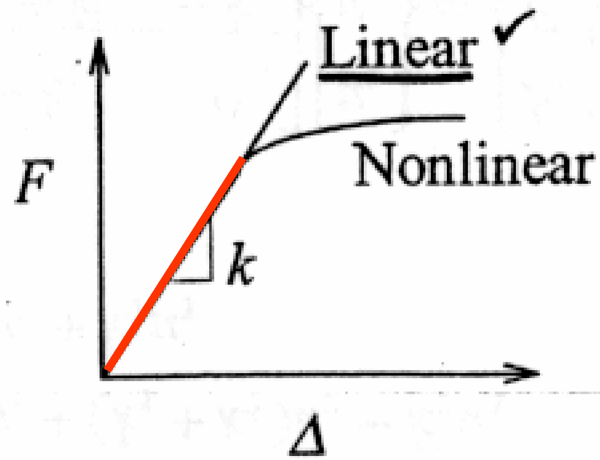
$$Kx = F$$



Simplest

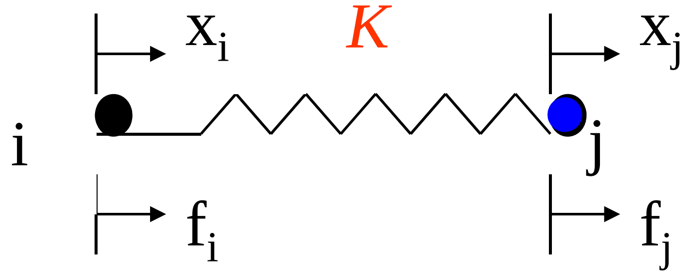
Spring force-displacement relationship:

$$F = k\Delta \qquad \text{with } \Delta = u_j - u_i$$



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

Stiffness Equation of One Spring

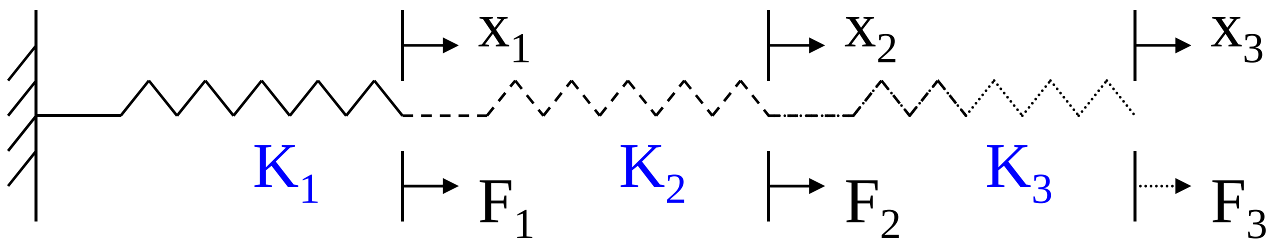


$$\begin{cases} K(x_i - x_j) = f_i \\ -K(x_i - x_j) = f_j \end{cases} \longrightarrow \begin{bmatrix} k_{ii} & k_{ij} \\ k_{ji} & k_{jj} \end{bmatrix} \begin{bmatrix} x_i \\ x_j \end{bmatrix} = \begin{bmatrix} f_i \\ f_j \end{bmatrix}$$

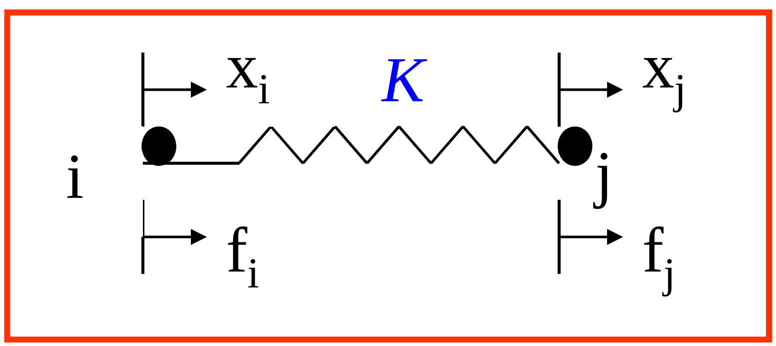
By using the unit displacement method, we can express the stiffness coefficients k_{ij} etc. in terms of the spring coefficient K

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

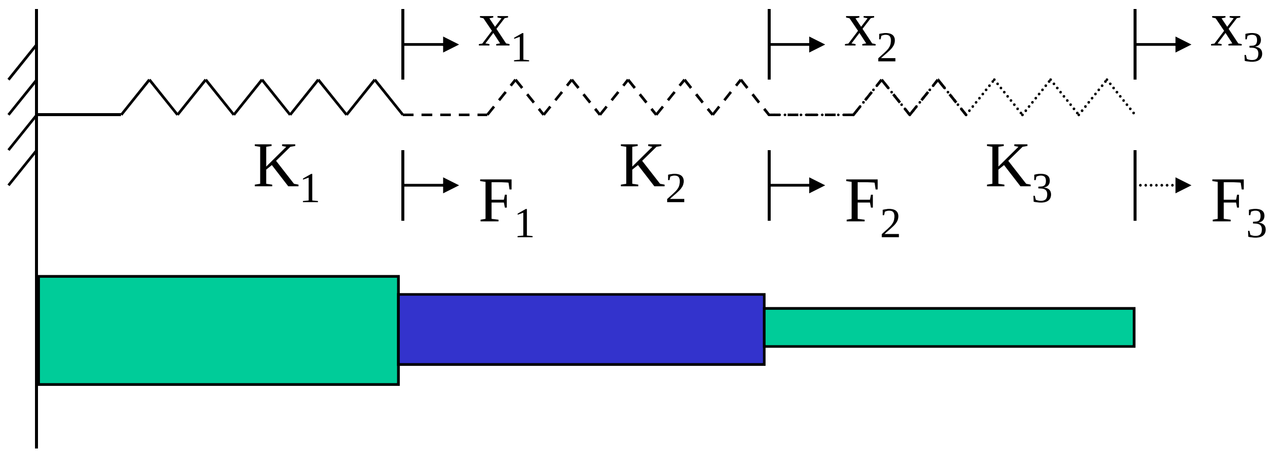
The Three-Spring System



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



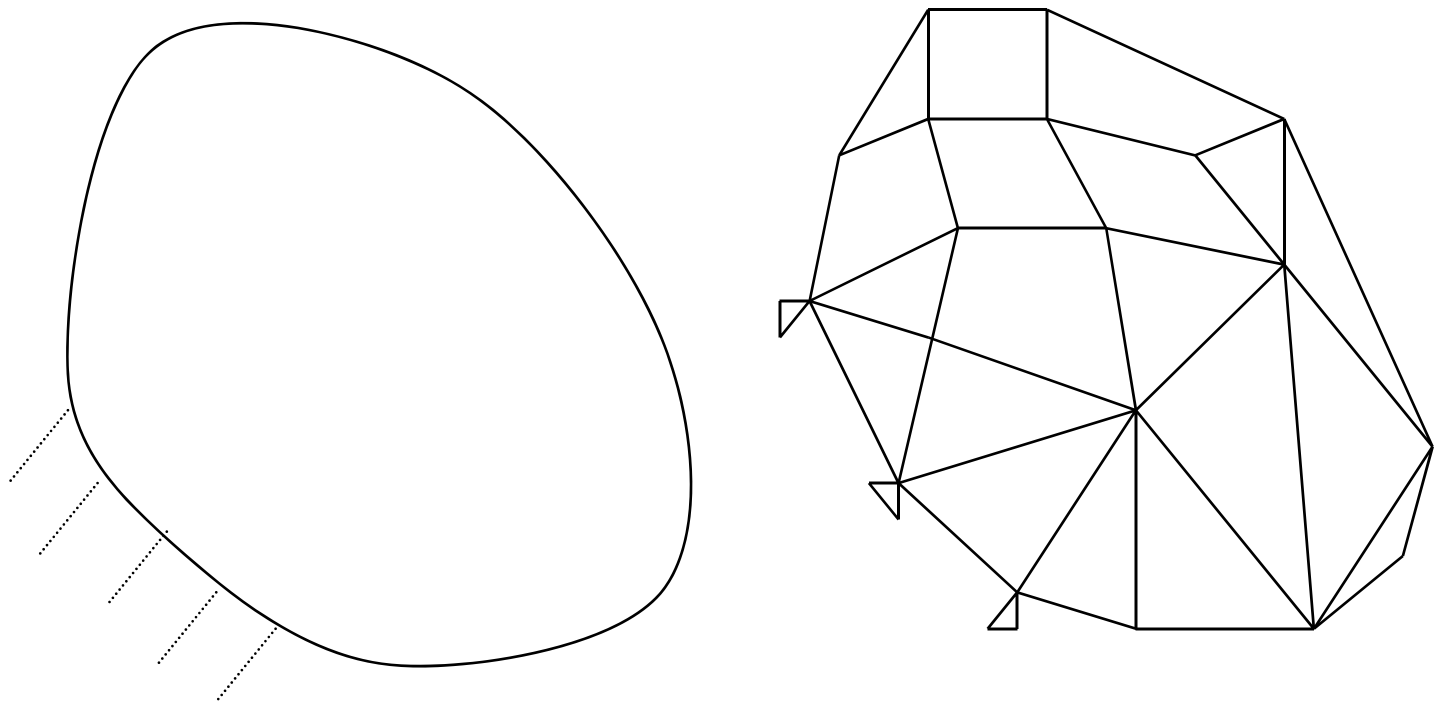
Identify and Solve the Stiffness Equations for a System of "Finite Elements"



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

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 <u>finite dimensions</u> - hence the word ***finite element***.
- The discrete "**equivalent spring**" system provides <u>an approximate model for the actual elastic body</u>
- 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.

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

$\boxed{\text{Real-Complex}}$

Procedure for Carrying out Finite Element Analysis

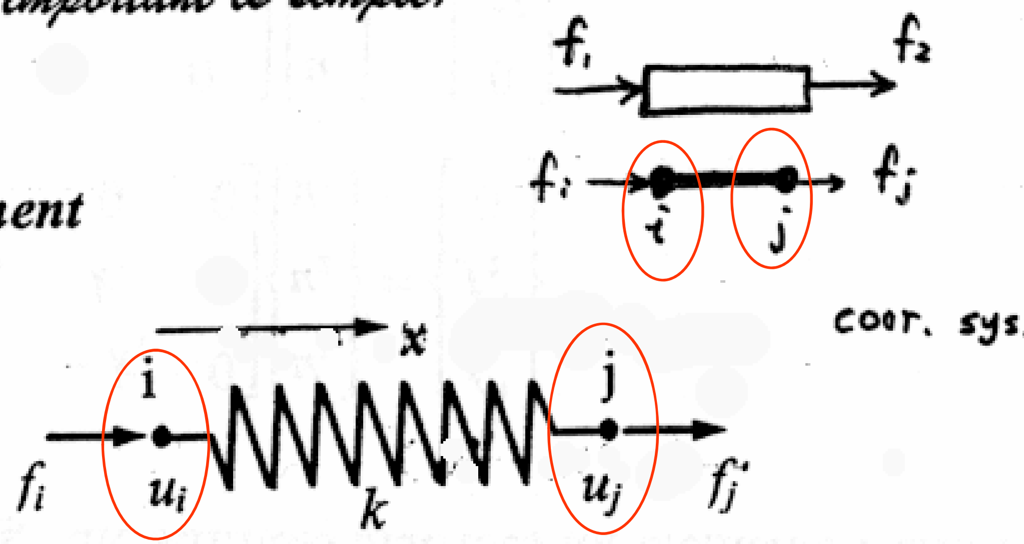
To construct the stiffness equations of a complex system made up of springs, one need to develop the **stiffness equation** of ***one spring*** and use the equation as a ***building block***

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

Spring Element

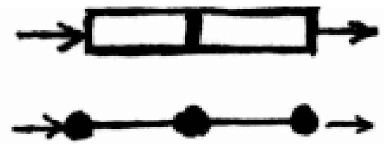
"Everything important is simple."

One Spring Element

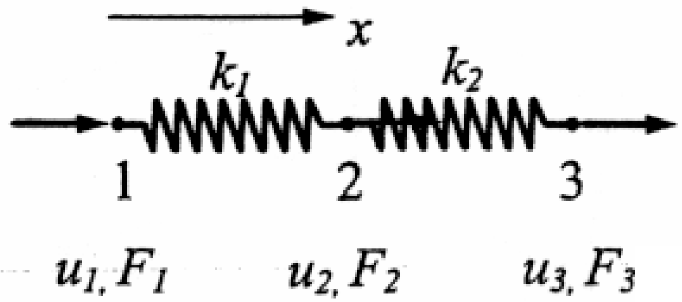


Two nodes:	i, j
Nodal displacements:	u_i, u_j (in, m, mm)
Nodal forces:	f_i, f_j (lb, Newton)
Spring constant (stiffness):	k (lb/in, N/m, N/mm)

Spring System

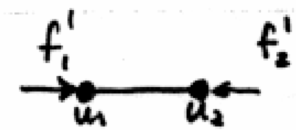


3 nodes
2 elements



For element 1,

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



$f_1^1 = k_1(u_1 - u_2)$

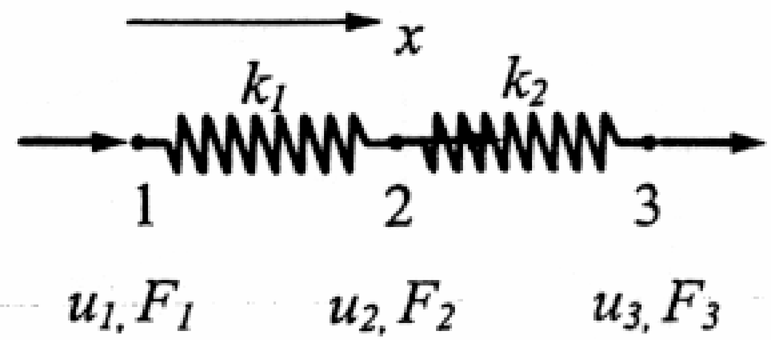
$-f_1^2 = k_1(u_1 - u_2)$

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* ($i = 1, 2$).

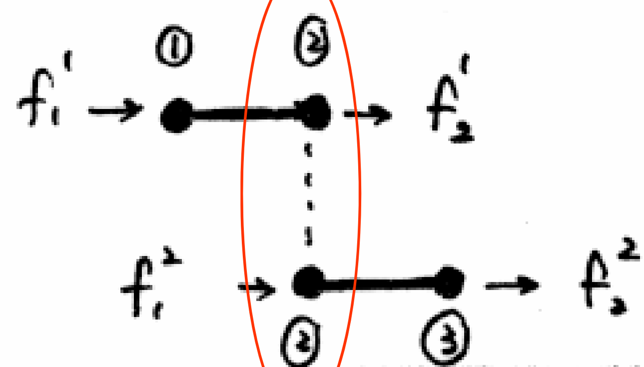
3 nodes
2 elements



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

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

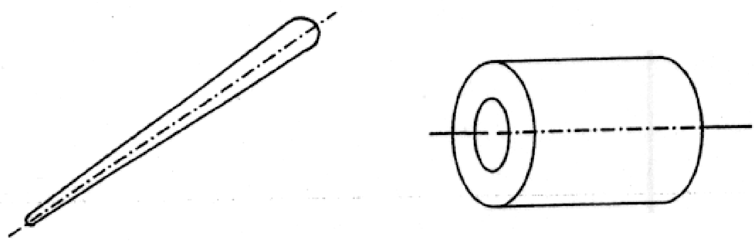
$$\begin{bmatrix} 0 & 0 & 0 \\ 0 & k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ f_1^2 \\ f_2^2 \end{Bmatrix}$$

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.**

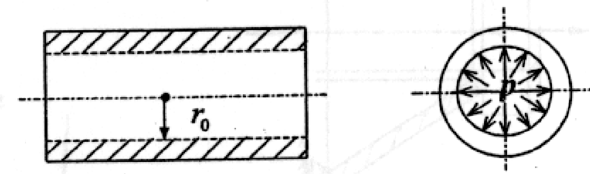
Applying Different Types of Loads

***Solids of Revolution* (Axisymmetric Solids):**



Baseball bat — *shaft*

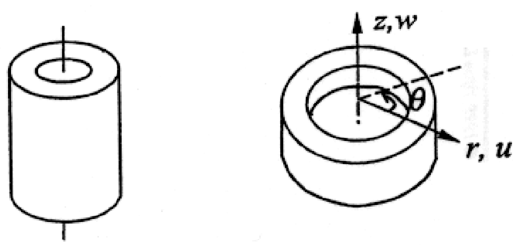
- ***Cylinder Subject to Internal Pressure:***



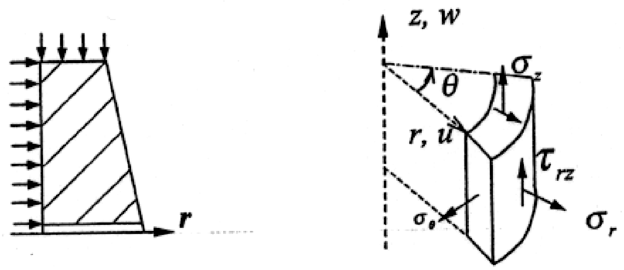
$q=(p)2\pi r_0$

Apply cylindrical coordinates:

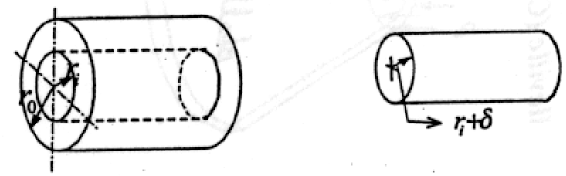
$(x, y, z) \Rightarrow (r, \theta, z)$



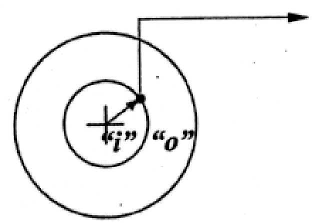
Evenly distributed load



- ***Press Fit:***



ring (Sleeve) — *shaft*



$at\ r=r_i:$

$u_o-u_i=\delta$

$\Rightarrow$ MPC

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

from CAD Model / FEM User Interface

simplified geometry

① ② ③

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

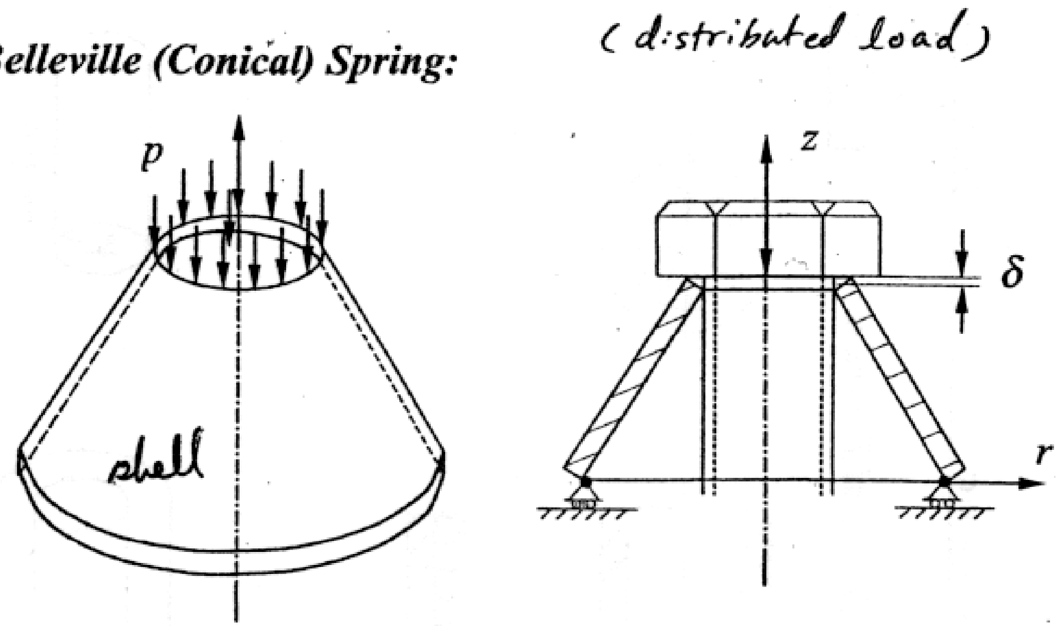
Advantages of General Purpose Programs

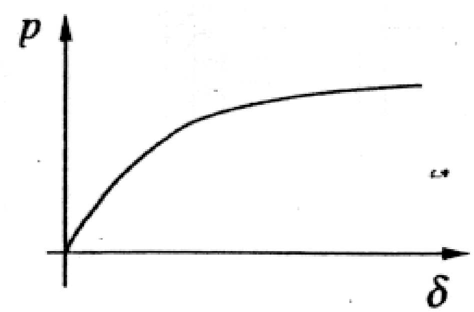
- Easy input - preprocessor
- Solves many types of problems
- Modular design - fluids, dynamics, heat, etc.
- Can run on PC’s now.
- Relatively low cost.

Limitations of Regular FEA Software

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

- ***Belleville (Conical) Spring:***





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