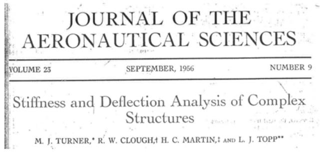
Advanced Computational Solid Mechanics

AES

General introduction


JOURNAL OF THE AERONAUTICAL SCIENCES

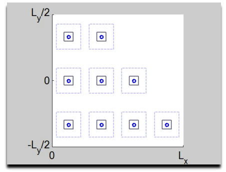
VOLUME 23 SEPTEMBER, 1956 NUMBER 9

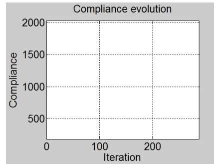
Stiffness and Deflection Analysis of Complex Structures

M. J. TURNER,* R. W. CLOUGH,† H. C. MARTIN,‡ AND L. J. TOPP**

About Me? http://institut-clement-ader.org/author/jmorlier/

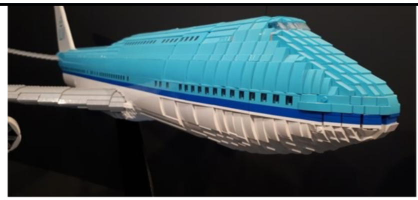
- Prof in Structural and Multidisciplinary Optimization
- Bat38 SUPAERO
- Research Lab (ICA)





Popularization

https://www.linkedin.com/pulse/possible-build-aircraft-wing-lego-joseph-morlier/?articleId=6627240732975480832



https://www.tripadvisor.fr/LocationPhotoDirectLink-g187529-d574612-i349532022-Museum_of_Natural_Science_Museo_de_Ciencias_Naturales-Valencia_Province_o.html

Is it possible to build an aircraft wing in LEGO® ?

Publié le 17 février 2020 Modifier l'article | Voir les stats

joseph morlier
Professor in Structural and Multidisciplinary Design Optimization, ... any idea? 3 articles

All the profs

Monsieur Joseph MORLIER	SUPAERO	Recap
Monsieur Michel MAHE	AIRBUS	Overview CSM Airbus
Monsieur Julien LE FANIC	AIRBUS	Vibration
Monsieur Simone CONIGLIO	AIRBUS	Topology optimization
Monsieur Julien RODES	AIRBUS	Nonlinear analysis, V&V
Madame Emmeline FAISSE	AIRBUS/SUPAERO	Exercices on RECAP
Monsieur Jean-François PINAZZA	AIRBUS DEFENSE AND SPACE	Practice of Nastran
Monsieur Frederic Lachaud	SUPAERO	Practice of Abaqus
Madame Christine Espinosa	SUPAERO	MultiPhysics

Contents

Courses:

- Introduction and basis of FEA*
- Vibration and transient analysis
- Nonlinear analysis stress, buckling
- Structural and Topology Optimization

Workshops:

- Airbus Overview on CSM
- FEA for Multiphysics applications
- V&V in FEA

Classes Exercices:

- Truss and beam formulation
- Matrix conditioning
- CST versus QUAD element
- Modeling Hypothesis for real life structures

Practice of Nastran:

- Learn the bdf in statics
- I beam Modeling in 1D,2D, 2.5D
- Modal analysis of a plate

Practice of Abaqus:

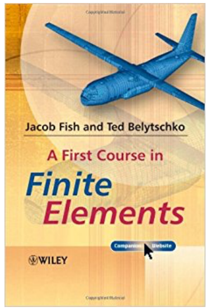
- Plate linear and nonlinear buckling

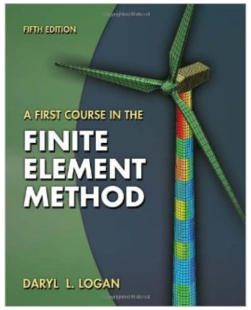
Practice of Hyper Works:

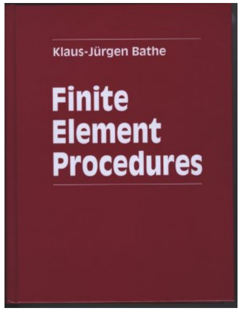
- From CAD TO FEA
- Topology Optimization in Optistruct
- Gradient computing

References

web.mit.edu/kjb/www/Books/FEP_2nd_Edition_4th_Printing.pdf







(more on the -draft- textbook)

The way we will work together

Basics of Scientific computing 2H with MATLAB
Course 0 with Exercice at IT center on 5/10
→finish livescripts until 13/10
Basics of Scientific computing 2H with Python
Course 0 with Exercice at IT center on 4/11

Basics of FEA 2H with MATLAB

- **Course 1** on 5/10 with a small exercice on 1D truss assembly
- **Course2** on 7/10 with a small miniproject on 2D membrane

Online courses review+ read lecture note until 13/10 to do in autonomy

https://github.com/jomorlier/afeacourse

Numerical Methods: Basis Choices

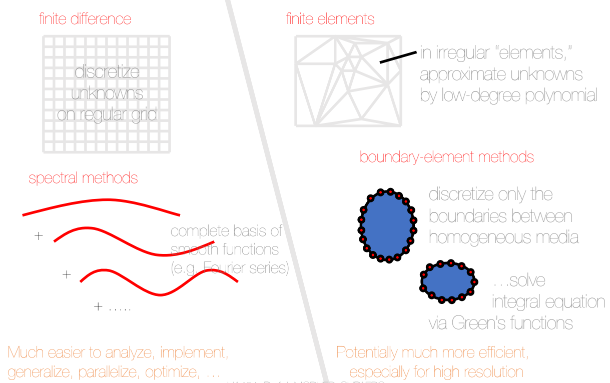
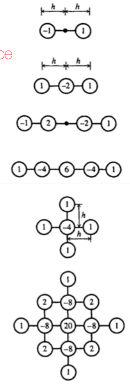


TABLE 3.1 *Finite difference approximations for various differentiations*

Differentiation	Finite difference approximation	Molecules
$\left.\frac{dw}{dx}\right\|_i$	$\frac{w_{i+1} - w_{i-1}}{2h}$	
$\left.\frac{d^2w}{dx^2}\right\|_i$	$\frac{w_{i+1} - 2w_i + w_{i-1}}{h^2}$	
$\left.\frac{d^3w}{dx^3}\right\|_i$	$\frac{w_{i+2} - 2w_{i+1} + 2w_{i-1} - w_{i-2}}{2h^3}$	
$\left.\frac{d^4w}{dx^4}\right\|_i$	$\frac{w_{i+2} - 4w_{i+1} + 6w_i - 4w_{i-1} + w_{i-2}}{h^4}$	
$\nabla^2 w\|_{i,j}$	$\frac{-4w_{i,j} + w_{i+1,j} + w_{i,j+1} + w_{i-1,j} + w_{i,j-1}}{h^2}$	
$\nabla^4 w\|_{i,j}$	$[20w_{i,j} - 8(w_{i+1,j} + w_{i-1,j} + w_{i,j+1} + w_{i,j-1}) + 2(w_{i+1,j+1} + w_{i-1,j+1} + w_{i-1,j-1} + w_{i+1,j-1}) + w_{i+2,j} + w_{i-2,j} + w_{i,j+2}$ $w_{i,j-2}]/h^4$	



Uniform spacing h; error in each case is $o(h^2)$. Point i or (i, j) is being considered; and $i \pm \cdots$ denotes points in the x-direction; $j \pm \cdots$ denotes points in the y-direction.

EXAMPLE 3.1: Figure E3.1 shows a system of three rigid carts on a horizontal plane that are interconnected by a system of linear elastic springs. Calculate the displacements of the carts and the forces in the springs for the loading shown.

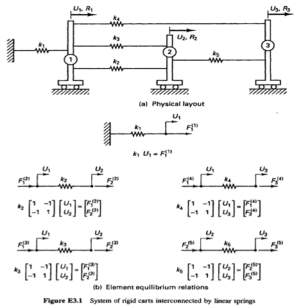


Figure E3.1 System of rigid carts interconnected by linear springs

$$\mathbf{KU} = \mathbf{R}$$

Direct Stiffness approach

where

$$\mathbf{U}^T = [U_1 \quad U_2 \quad U_3]$$

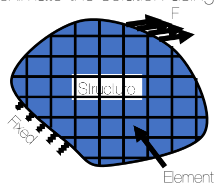
$$\mathbf{K} = \begin{bmatrix} (k_1 + k_2 + k_3 + k_4) & -(k_2 + k_3) & -k_4 \\ -(k_2 + k_3) & (k_2 + k_3 + k_5) & -k_5 \\ -k_4 & -k_5 & (k_4 + k_5) \end{bmatrix}$$

and

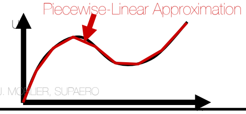
$$\mathbf{R}^T = [R_1 \quad R_2 \quad R_3]$$

INTRODUCTION TO FINITE ELEMENT

- What is the finite element method (FEM)?
  - A technique for obtaining approximate solutions to boundary value problems.
  - Partition of the domain into a set of simple shapes (element)
  - Approximate the solution using piecewise polynomials within the element

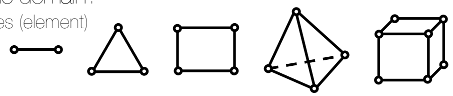


$$\begin{cases} \frac{\partial \sigma_{xx}}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + f_{vx} = 0 \\ \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \sigma_{yy}}{\partial y} + f_{vy} = 0 \end{cases} \qquad \underline{div}(\underline{\underline{\sigma}}) + \underline{f}_v = \underline{0}$$

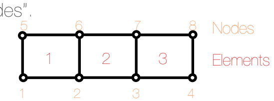


INTRODUCTION TO FEM *cont.*

- How to discretize the domain?
  - Using simple shapes (element)

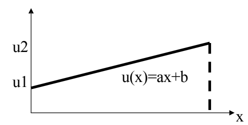


- All elements are connected using "nodes".

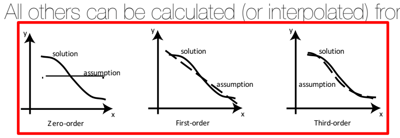


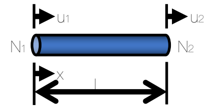
- Solution at Element 1 is described using the values at Nodes 1, 2, 6, and 5 (Interpolation).
- Elements 1 and 2 share the solution at Nodes 2 and 6.

INTRODUCTION TO FEM *cont.*



- Finite element analysis solves for nodal values.
  - All others can be calculated (or interpolated) from nodal solutions





- Displacement within the element

$$u(x) = a + bx = u_1 + \frac{u_2 - u_1}{L}x = \frac{L-x}{L}u_1 + \frac{x}{L}u_2$$

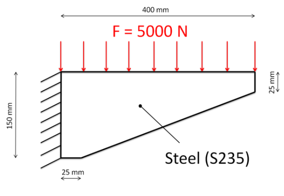
- Strain of the element

Interpolation (Shape) Function

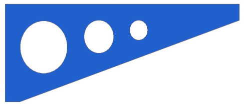
$$\varepsilon(x) = \frac{\partial u}{\partial x} = -\frac{1}{L}u_1 + \frac{1}{L}u_2$$

Standard Engineer

- The problem is the following: We want to find the lightest component that can withstand 5000 N.
- The component is made of steel (S235) and the maximum dimensions are shown here:

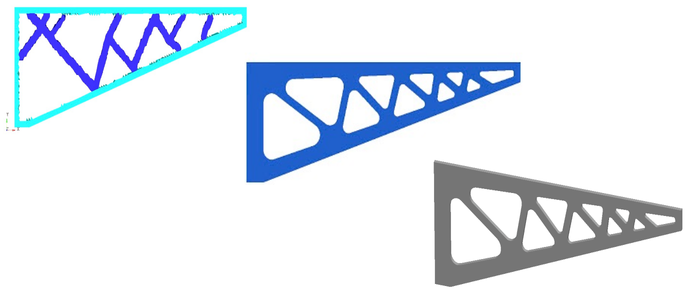


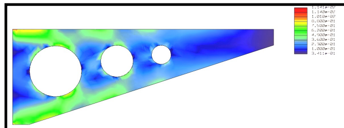
Thickness = 5 mm

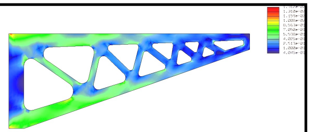


- If you ask an experienced Mechanical Engineer, he would say: Make some holes!

SUPAERO's Engineers use Topopt







- Steel S235 has a yield strength of 235 MPa (for 5 mm thickness). Using a **Safety Factor** of 1.5, maximum von Mises stress should be below 155 MPa. Both cases are below that level (with holes: 110 MPa, optimized: 130 MPa). Maximum displacements are around 0.4 mm in both cases.

- A Topology Optimization maximizes the stiffness of a component applying a volume constraint (reducing 40% in volume for example). →you get a very **lightweight** solution which is also very **Stiff**

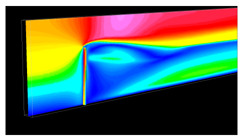
- You can also see that the stress distribution in the model optimized is much diffused than in the model with holes. Why?

→ Because most of the component is under stress**, leaving no area unstressed**. This means material being used only where it's going to be necessary (under stress). → Smart use of material!

Objectives of This FEM Course

1. Understand the fundamental ideas of the FEM
2. Know the behavior and usage of each type of elements covered in this course
3. Be able to prepare a suitable FE model for given problems
4. Can interpret and evaluate the quality of the results (know the physics of the problems)
5. Be aware of the limitations of the FEM (don't misuse the FEM - a numerical tool)

Never(!) trust colorful pictures



With Me

4 courses to review online

- 2 Matlab assignements* (#1,#4), publish reports on LMS.
- Slides to read in autonomy
- Link with the Class Exercices

With My PhDs

2 exercices (2h) + review online correction

More Info on LMS

With Engineers from industry and lab

S. Coniglio J. rodes (Airbus), JF. Pinazza (AIRBUS DS),

Computer labs on Patran-Nastran, Abaqus and Optistruct

Several projects will be marked*

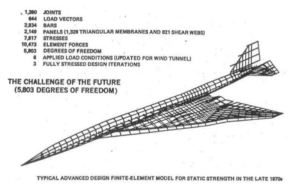
Linear analysis

- Most structural analysis problems can be treated as linear static problems, based on the following assumptions
- **Small deformations** (loading pattern is not changed due to the deformed shape)
- **Elastic materials** (no plasticity or failures)
- **Static loads** (the load is applied to the structure in a slow or steady fashion)
- **Modal Analysis** (vibration with or without external forces)
- Linear analysis can provide most of the information about the behavior of a structure, and can be a good approximation for many analyses.

History (see LMS)

A Brief History of the FEM

- 1943 ----- Courant (Variational methods)
- 1956 ----- Turner, Clough, Martin and Topp (Stiffness)
- 1960 ----- Clough ("Finite Element", plane problems)
- 1970s ----- Applications on mainframe computers
- 1980s ----- Microcomputers, pre- and postprocessors
- 1990s ----- Analysis of large structural systems *
- NOW? See later …



TYPICAL ADVANCED DESIGN FINITE-ELEMENT MODEL FOR STATIC STRENGTH IN THE LATE 1970s

What FE can do (0D+1D)

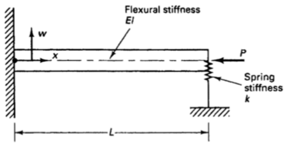


Figure E3.20 Column subjected to a compressive load

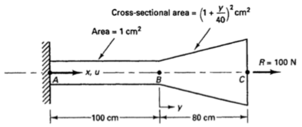


Figure E3.22 Bar subjected to a concentrated end force

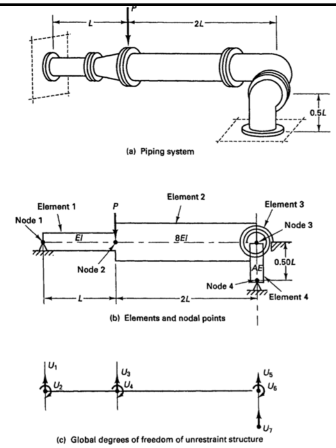


Figure E4.1 Piping system and idealization

What FE can do (2D+1D)

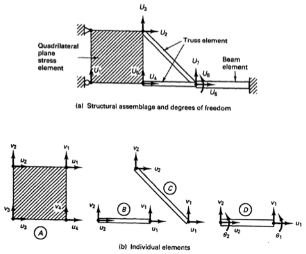
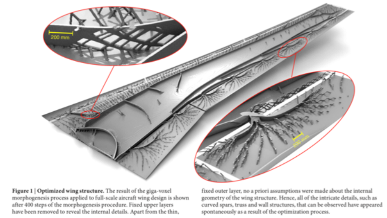


Figure E4.11 A simple element assemblage



LETTER

doi:10.1038/nature23911

Giga-voxel computational morphogenesis for structural design

Niels Aage[1,2], Erik Andreassen[1], Boyan S. Lazarov[1] & Ole Sigmund[1]

what do all of these equations have in common?

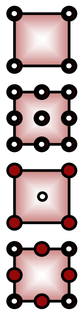
$$\frac{\mathrm{d}\sigma(x)}{\mathrm{d}x} + f(x) = 0 \qquad \sigma(x) = E\,\epsilon(x) \qquad \epsilon(x) = \frac{\mathrm{d}u(x)}{\mathrm{d}x}$$

they are second order partial differential equations!

$$E\,\frac{\mathrm{d}^2u(x)}{\mathrm{d}x^2} + f(x) = 0$$

the finite element method loves second order pde's!

steps of a finite element simulation



#1 modeling (boundary value problem)

$$\mathrm{div}\boldsymbol{\sigma} + \boldsymbol{f} = \mathbf{0}$$

#2 pre-processing (discretization with finite elements)

#3 algebraization (system of equations)

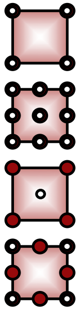
$$\mathbf{K}\cdot\mathbf{u} = \mathbf{F}$$

#4 solution $\mathbf{u} = \mathbf{K}^{-1}\cdot\mathbf{F}$

#5 post-processing (final calculation and visualization)

$$\boldsymbol{\sigma} = \mathbf{E}\cdot\boldsymbol{\epsilon}(\boldsymbol{u})$$

... and what can go wrong...



#1 modeling (boundary value problem) — modeling error (wrong equations)

$$\mathrm{div}\boldsymbol{\sigma} + \boldsymbol{f} = \mathbf{0}$$

#2 pre-processing (discretization with finite elements) — discretization error (wrong elements)

#3 algebraization (system of equations)

$$\mathbf{K} \cdot \mathbf{u} = \mathbf{F}$$

#4 solution — solution error (wrong algorithms)

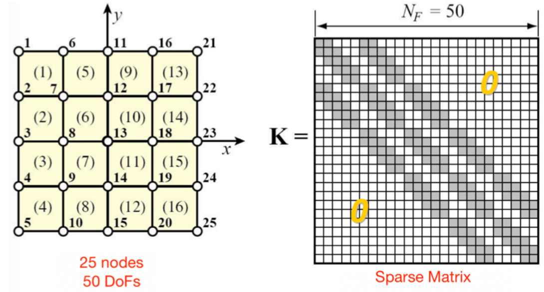
$$\mathbf{u} = \mathbf{K}^{-1} \cdot \mathbf{F}$$

#5 post-processing (final calculation and visualization) — visualization error (wrong colorscales)

$$\boldsymbol{\sigma} = \mathbf{E} \cdot \boldsymbol{\epsilon}(\boldsymbol{u})$$

Sparse?

- See on LMS HPC Supplementary materials



Computer Implementations

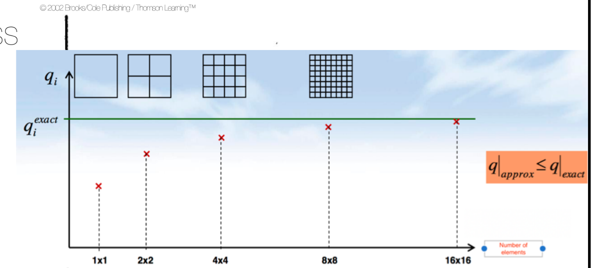
- Preprocessing (build FE model, loads and constraints)
- FEA solver (assemble and solve the system of equations)
- Postprocessing (sort and display the results)

It's up to you

Academic project: We know the solution

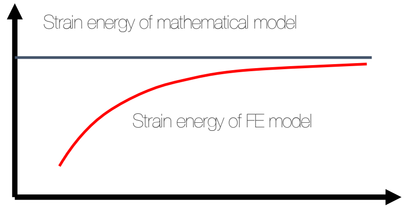
Criteria for monotonic convergence

1. COMPLETENESS
2. COMPATIBILITY

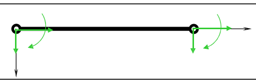
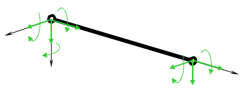


Important property of finite element solution:

When the conditions of monotonic convergence are satisfied (compatibility and completeness) the finite element strain energy always underestimates the strain energy of the actual structure



1D Element types

	Name	DOF per Node	GEOMETRY
1 dimension	Bar	u	
	Beam	u v α	
	Spatial Beam	u v w α β γ	

BTW…

Why do you have to input (in commercial software …such as Nastran) for bar element:

- A
- E
- Nu
- L
- but also a **torsional factor J** ???

$$K_{e,axial} = \frac{EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \qquad K_{e,torsion} = \frac{GJ}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

What about G …

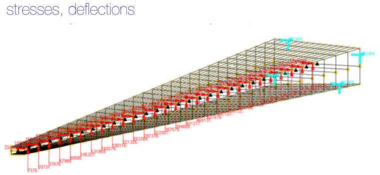
→Remember Linear Elasticity E=2(1+nu)G

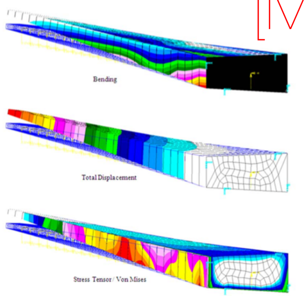
K, M???

We need accurate static analysis of complex structures to estimate stresses, deflections

Dynamic analysis :=natural frequencies

[M]





F_i=[K]U_i

After we have applied loads, we need to create analytical models that provide input/output relationships between applied loads and internal forces, moments and stresses

K properties

- The stiffness matrix of a structure is called the matrix K making it possible to express the deformation energy in a quadratic form of the displacements.

$$E_{def} = \frac{1}{2} U^T K U$$

- The eigenvalues of the stiffness matrix are obtained by solving the problem:

We can write $K\Phi_i = \lambda_i \Phi_i$ — λ : i[th] eigenvalue, Φ_i : i[th] eigenvector

$$\lambda_i = \frac{\Phi_i^T K \Phi_i}{\Phi_i^T \Phi_i}$$ if Φ_i respect: $\Phi_i^T \Phi_i = 1$

$$\lambda_i = \Phi_i^T K \Phi_i = 2E_{def}$$

The eigenvalues of a stiffness matrix represent (proportionaly to a coefficient) the total strain energy involved by the deformation modes of the structure.

K Properties (2)

- Case of free structures (or with mechanism): In this case, there are a number of zero eigenvalues (3 for 2D problems, 6 for 3D problems).
- They correspond to general displacement modes for which the strain energy is zero.--> They are called rigid body modes or rigid modes.
- The rigidity matrix of a free structure is therefore semi-positive
- Example: bar in tension - compression

$$\det\begin{pmatrix} \frac{ES}{L}-\lambda & \frac{-ES}{L} \\ \frac{-ES}{L} & \frac{ES}{L}-\lambda \end{pmatrix} = 0$$

Rigid body mode

$$\lambda = 0 \quad \Phi = \begin{pmatrix} 1 \\ 1 \end{pmatrix}$$

Pure compression mode

$$\lambda = \frac{2ES}{L} \quad \Phi = \begin{pmatrix} 1 \\ -1 \end{pmatrix}$$

BCs

Taking into account the prescribed displacement conditions, 3 possibilities:

Penalisation method — Application of a digital "weight" on Coefficients of the stiffness matrix

Lagrange Multipliers — The system of equation (KU = F) is completed by constraint equations

The Partition Method

The Partition Method

- The system of equations of equilibrium is constituted without taking into account the conditions of prescribed displacements.

$$K_G U_G = F_G$$

- We obtain a system of the form: The displacement vector is decomposed (partition) as follows:

$$U_G = \begin{pmatrix} U_a \\ U_b \end{pmatrix}$$

: free displacements (unknown)

: Prescribed displacements (known!!!)

- This partition is applied to the loading vector and the stiffness matrix

$$F_G = \begin{pmatrix} F_a \\ F_b \end{pmatrix}$$

: forces corresponding to free displacements (known!!)

: forces corresponding to prescribed displacements (unknown) Reactions

$$K_G = \begin{pmatrix} K_{aa} & K_{ab} \\ K_{ba} & K_{bb} \end{pmatrix}$$

The Partition Method

By developing equations of equilibrium, one obtains:

$$\begin{pmatrix} K_{aa} & K_{ab} \\ K_{ba} & K_{bb} \end{pmatrix}\begin{pmatrix} U_a \\ U_b \end{pmatrix} = \begin{pmatrix} F_a \\ F_b = R \end{pmatrix} \Leftrightarrow \begin{cases} K_{aa}U_a + K_{ab}U_b = F_a \\ K_{ba}U_a + K_{bb}U_b = R \end{cases}$$

➢ The first system of equations makes it possible to obtain the free displacements (Ua):

$$K_{aa}U_a = F_a - K_{ab}U_b \quad \text{or} \quad U_a = K_{aa}^{-1}(F_a - K_{ab}U_b)$$

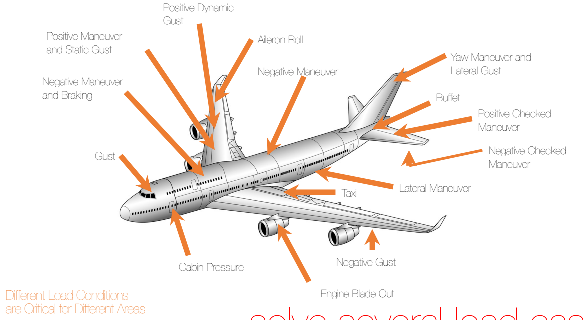
➢ The free displacements being known, the reactions are obtained with the second system of equations

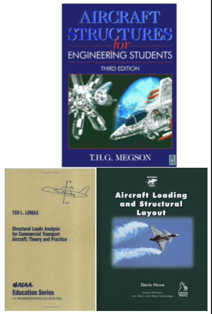
$$R = K_{ba}U_a + K_{bb}U_b$$

✓ Special case: ALL imposed displacements are zero (Ub = 0)

$$K_{aa}U_a = F_a \quad \text{ou} \quad U_a = K_{aa}^{-1}F_a \qquad R = K_{ba}U_a$$

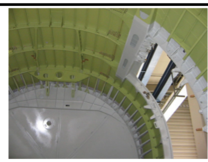
Inverse K ONLY once

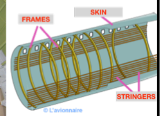




solve several load cases U_i=K-1*F_i

Fuselage in dynamics





Study of the dynamic characteristics of a structure:

- natural frequencies
- normal modes (shapes)

Let **f**(t) = **0** and **C** = **0** (ignore damping) in the dynamic equation (8) and obtain

$$\mathbf{M}\ddot{\mathbf{u}} + \mathbf{K}\mathbf{u} = \mathbf{0}$$

Assume that displacements vary harmonically with time, that is,

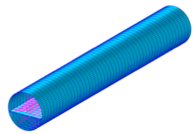
$$\mathbf{u}(t) = \bar{\mathbf{u}}\sin(\omega t),$$
$$\dot{\mathbf{u}}(t) = \omega\bar{\mathbf{u}}\cos(\omega t),$$
$$\ddot{\mathbf{u}}(t) = -\omega^2\bar{\mathbf{u}}\sin(\omega t),$$

where $\bar{\mathbf{u}}$ is the vector of nodal displacement amplitudes.

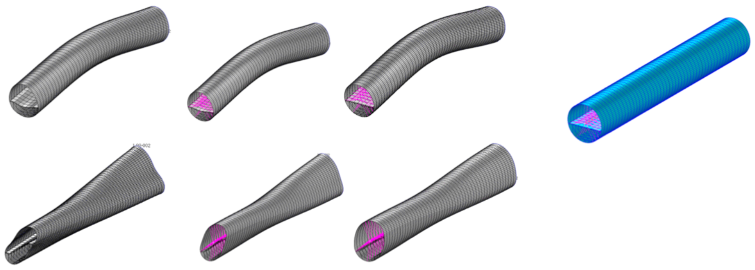
Eq. (12) yields,

$$\left[\mathbf{K} - \omega^2\mathbf{M}\right]\bar{\mathbf{u}} = \mathbf{0}$$

This is a generalized eigenvalue problem (EVP).



Fuselage in dynamics



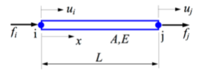
Let's start constructing [K] for several elements…

Plan

1. Bar FE theory + Example
2. Beam FE theory + Examples (see Course Exercice)
3. Local to Global coordinate + Example*
4. For [M] construction, see online materials

Stiffness Matrix --- A Formal Approach

We derive the same stiffness matrix for the bar using a formal approach which can be applied to many other more complicated situations.



L	length
A	cross-sectional area
E	elastic modulus
$u = u(x)$	displacement
$\varepsilon = \varepsilon(x)$	strain
$\sigma = \sigma(x)$	stress

$$u(x) = \left(1 - \frac{x}{L}\right)u_i + \frac{x}{L}u_j \qquad (1)$$

Strain-displacement relation:

$$\varepsilon = \frac{du}{dx} \qquad (2)$$

Define two linear shape functions as follows

$$N_i(\xi) = 1 - \xi, \qquad N_j(\xi) = \xi$$

where $\xi = \frac{x}{L}, \qquad 0 \le \xi \le 1$

From (1) we can write the displacement as

$$u(x) = u(\xi) = N_i(\xi)u_i + N_j(\xi)u_j$$

or

$$u = \begin{bmatrix} N_i & N_j \end{bmatrix} \begin{Bmatrix} u_i \\ u_j \end{Bmatrix} = \mathbf{Nu} \qquad (3)$$

Strain is given by (2) and (3) as

$$\varepsilon = \frac{du}{dx} = \left[\frac{d}{dx}\mathbf{N}\right]\mathbf{u} = \mathbf{Bu}$$

where B is the element strain-displacement matrix, which is

B matrix

Use chain rule

$$\mathbf{B} = \frac{d}{dx}\begin{bmatrix} N_i(\xi) & N_j(\xi) \end{bmatrix} = \frac{d}{d\xi}\begin{bmatrix} N_i(\xi) & N_j(\xi) \end{bmatrix} \bullet \frac{d\xi}{dx}$$

i.e., $\mathbf{B} = \begin{bmatrix} -1/L & 1/L \end{bmatrix}$ (constant here ? right)

Stress can be written as

$$\sigma = E\varepsilon \;\; = E\mathbf{Bu}$$

Consider the strain energy stored in the bar

$$U = \frac{1}{2}\int_V \sigma^{\mathrm{T}} \varepsilon dV = \frac{1}{2}\int_V (\mathbf{u}^{\mathrm{T}}\mathbf{B}^{\mathrm{T}} E\mathbf{Bu})dV$$

$$= \frac{1}{2}\mathbf{u}^{\mathrm{T}}\left[\int_V (\mathbf{B}^{\mathrm{T}} E\mathbf{B})dV\right]\mathbf{u}$$

B matrix to K (Finally…)

The work done by the two nodal forces is

$$W = \frac{1}{2} f_i u_i + \frac{1}{2} f_j u_j = \frac{1}{2}\mathbf{u}^{\mathrm{T}}\mathbf{f}$$

For conservative system, we stat that U=W

So:

$$\frac{1}{2}\mathbf{u}^{\mathrm{T}}\left[\int_V (\mathbf{B}^{\mathrm{T}} E\mathbf{B})dV\right]\mathbf{u} = \frac{1}{2}\mathbf{u}^{\mathrm{T}}\mathbf{f}$$

We can conclude that: $\left[\int_V (\mathbf{B}^{\mathrm{T}} E\mathbf{B})dV\right]\mathbf{u} = \mathbf{f}$

$$\boxed{\begin{gathered}\mathbf{ku} = \mathbf{f} \\ \text{where} \\ \mathbf{k} = \int_V (\mathbf{B}^{\mathrm{T}} E\mathbf{B})dV\end{gathered}}$$

K

V is A*L
integral over V is A*integral over L

K =

$$A\int B^t E\, B \mathrm{d}X$$

```
K = A* int(transpose(B) *E* B, X, sym(0), L)
```

K =

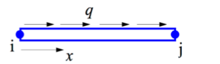
$$\begin{pmatrix} \frac{AE}{L} & -\frac{AE}{L} \\ -\frac{AE}{L} & \frac{AE}{L} \end{pmatrix}$$

- K:= $\frac{EA}{L}\begin{pmatrix} +1 & -1 \\ -1 & +1 \end{pmatrix}$ is called the stiffness matrix
- Relates Force to Displacement of each DOF
- Semi Definite Positive & Symmetric

Distributed load

- Uniformly distributed axial load q (N/mm, N/m, lb/in) can be converted to two equivalent nodal forces of magnitude qL/2.

-->We verify this by considering the work done by the load q,



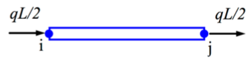
$$W_q = \int_0^L \frac{1}{2} uq dx = \frac{1}{2}\int_0^1 u(\xi) q(L d\xi) = \frac{qL}{2}\int_0^1 u(\xi) d\xi$$

$$= \frac{qL}{2}\int_0^1 \begin{bmatrix} N_i(\xi) & N_j(\xi) \end{bmatrix}\begin{Bmatrix} u_i \\ u_j \end{Bmatrix} d\xi$$

$$= \frac{qL}{2}\int_0^1 \begin{bmatrix} 1-\xi & \xi \end{bmatrix} d\xi \begin{Bmatrix} u_i \\ u_j \end{Bmatrix}$$

$$= \frac{1}{2}\begin{bmatrix} \frac{qL}{2} & \frac{qL}{2} \end{bmatrix}\begin{Bmatrix} u_i \\ u_j \end{Bmatrix}$$

$$= \frac{1}{2}\begin{bmatrix} u_i & u_j \end{bmatrix}\begin{Bmatrix} qL/2 \\ qL/2 \end{Bmatrix}$$



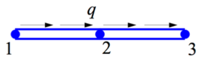
Distributed load

That is: $W_q = \frac{1}{2}\mathbf{u}^T\mathbf{f}_q$ with $\mathbf{f}_q = \begin{Bmatrix} qL/2 \\ qL/2 \end{Bmatrix}$

Thus, from the U=W concept for the element, we have

$$\frac{1}{2}\mathbf{u}^T\mathbf{k}\mathbf{u} = \frac{1}{2}\mathbf{u}^T\mathbf{f} + \frac{1}{2}\mathbf{u}^T\mathbf{f}_q$$

In an assembly of bars,

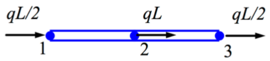


This leads to

$$\mathbf{k}\mathbf{u} = \mathbf{f} + \mathbf{f}_q$$

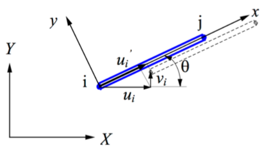
The new nodal force vector is

$$\mathbf{f} + \mathbf{f}_q = \begin{Bmatrix} f_i + qL/2 \\ f_j + qL/2 \end{Bmatrix}$$



But… in 2D & 3D space?

Transformation



Transformation

$$u_i' = u_i \cos\theta + v_i \sin\theta = \begin{bmatrix} l & m \end{bmatrix}\begin{Bmatrix} u_i \\ v_i \end{Bmatrix}$$

$$v_i' = -u_i \sin\theta + v_i \cos\theta = \begin{bmatrix} -m & l \end{bmatrix}\begin{Bmatrix} u_i \\ v_i \end{Bmatrix}$$

$$\boxed{\text{where } l = \cos\theta,\ m = \sin\theta.}$$

Local	*Global*
x, y	X, Y
u_i', v_i'	u_i, v_i
1 dof at a node	2 dof's at a node

In matrix form $\begin{Bmatrix} u_i' \\ v_i' \end{Bmatrix} = \begin{bmatrix} l & m \\ -m & l \end{bmatrix}\begin{Bmatrix} u_i \\ v_i \end{Bmatrix}$

In compact form $\mathbf{u}_i' = \tilde{\mathbf{T}}\mathbf{u}_i$

Note: Lateral displacement v_i' does not contribute to the stretch of the bar, within the linear theory.

Transformation matrix

$$\tilde{\mathbf{T}} = \begin{bmatrix} l & m \\ -m & l \end{bmatrix}$$

$$\boxed{\text{where } l = \cos\theta,\ m = \sin\theta.}$$

is orthogonal, that is, $\tilde{\mathbf{T}}^{-1} = \tilde{\mathbf{T}}^T$

For the two nodes of the bar element, we have

$$\begin{Bmatrix} u_i' \\ v_i' \\ u_j' \\ v_j' \end{Bmatrix} = \begin{bmatrix} l & m & 0 & 0 \\ -m & l & 0 & 0 \\ 0 & 0 & l & m \\ 0 & 0 & -m & l \end{bmatrix}\begin{Bmatrix} u_i \\ v_i \\ u_j \\ v_j \end{Bmatrix}$$

$$\boxed{\mathbf{u}' = \mathbf{T}\mathbf{u} \qquad \text{with } \mathbf{T} = \begin{bmatrix} \tilde{\mathbf{T}} & \mathbf{0} \\ \mathbf{0} & \tilde{\mathbf{T}} \end{bmatrix}}$$

$$\boxed{\mathbf{f}' = \mathbf{T}\mathbf{f}}$$

The nodal forces are transformed in the same way,

Stiffness Matrix in the 2-D Space

In the local coordinate system, we have

$$\frac{EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} u_i' \\ u_j' \end{Bmatrix} = \begin{Bmatrix} f_i' \\ f_j' \end{Bmatrix}$$

augmenting leads to

$$\frac{EA}{L}\begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}\begin{Bmatrix} u_i' \\ v_i' \\ u_j' \\ v_j' \end{Bmatrix} = \begin{Bmatrix} f_i' \\ 0 \\ f_j' \\ 0 \end{Bmatrix}$$

Or $\mathbf{k'u'} = \mathbf{f'}$ > $\mathbf{k'Tu} = \mathbf{Tf}$

Multiplying both sides by $\mathbf{T}^T$ and noticing that $\mathbf{T}^T\mathbf{T} = \mathbf{I}$, we obtain

Thus, the element stiffness matrix $\mathbf{k}$ in the global coordinate system is

$\mathbf{k} = \mathbf{T}^T\mathbf{k'T}$ which is a 4x4 symmetric matrix.

Explicit form

$$\mathbf{k} = \frac{EA}{L}\begin{array}{c} \begin{array}{cccc} u_i & v_i & u_j & v_j \end{array} \\ \begin{bmatrix} l^2 & lm & -l^2 & -lm \\ lm & m^2 & -lm & -m^2 \\ -l^2 & -lm & l^2 & lm \\ -lm & -m^2 & lm & m^2 \end{bmatrix} \end{array}$$

Calculation of the directional cosines l and m:

$$l = \cos\theta = \frac{X_j - X_i}{L}, \qquad m = \sin\theta = \frac{Y_j - Y_i}{L}$$

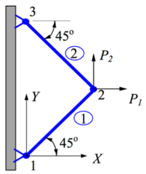
The structure stiffness matrix is assembled by using the element stiffness matrices in the usual way as in the 1-D case.

Example (bar 2D)

A simple plane truss is made of two identical bars (with E, A, and L), and loaded as shown in the figure. Find

1) displacement of node 2;

2) stress in each bar.

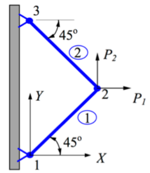


Example 2 (bar)

A simple plane truss is made of two identical bars (with E, A, and L), and loaded as shown in the figure. Find

1) displacement of node 2;

2) stress in each bar.



In local coordinate systems, we have

$$\mathbf{k}_1' = \frac{EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = \mathbf{k}_2'$$

These two matrices cannot be assembled together, because they are in different coordinate systems. We need to convert them to the global coordinate system OXY.

For element 1: $\theta = 45°$, $l = m = \frac{\sqrt{2}}{2}$ >

$$\mathbf{k}_1 = \mathbf{T}_1^T\mathbf{k}_1'\mathbf{T}_1 = \frac{EA}{2L}\begin{array}{c} \begin{array}{cccc} u_1 & v_1 & u_2 & v_2 \end{array} \\ \begin{bmatrix} 1 & 1 & -1 & -1 \\ 1 & 1 & -1 & -1 \\ -1 & -1 & 1 & 1 \\ -1 & -1 & 1 & 1 \end{bmatrix} \end{array}$$

For element 2 : $\theta = 135°$, $l = -\frac{\sqrt{2}}{2}$, $m = \frac{\sqrt{2}}{2}$ >

$$\mathbf{k}_2 = \mathbf{T}_2^T\mathbf{k}_2'\mathbf{T}_2 = \frac{EA}{2L}\begin{array}{c} \begin{array}{cccc} u_2 & v_2 & u_3 & v_3 \end{array} \\ \begin{bmatrix} 1 & -1 & -1 & 1 \\ -1 & 1 & 1 & -1 \\ -1 & 1 & 1 & -1 \\ 1 & -1 & -1 & 1 \end{bmatrix} \end{array}$$

Assemble & solve & postprocess

$$
\frac{EA}{2L}
\begin{array}{c}
\begin{array}{cccccc} u_1 & v_1 & u_2 & v_2 & u_3 & v_3 \end{array} \\
\begin{bmatrix}
1 & 1 & -1 & -1 & 0 & 0 \\
1 & 1 & -1 & -1 & 0 & 0 \\
-1 & -1 & 2 & 0 & -1 & 1 \\
-1 & -1 & 0 & 2 & 1 & -1 \\
0 & 0 & -1 & 1 & 1 & -1 \\
0 & 0 & 1 & -1 & -1 & 1
\end{bmatrix}
\end{array}
\begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{Bmatrix}
=
\begin{Bmatrix} F_{1X} \\ F_{1Y} \\ F_{2X} \\ F_{2Y} \\ F_{3X} \\ F_{3Y} \end{Bmatrix}
$$

$$
\sigma = E\varepsilon = E\mathbf{B}\begin{Bmatrix} u_i' \\ u_j' \end{Bmatrix} = E\begin{bmatrix} -\frac{1}{L} & \frac{1}{L} \end{bmatrix}\begin{bmatrix} l & m & 0 & 0 \\ 0 & 0 & l & m \end{bmatrix}\begin{Bmatrix} u_i \\ v_i \\ u_j \\ v_j \end{Bmatrix}
$$

$$
\sigma = \frac{E}{L}\begin{bmatrix} -l & -m & l & m \end{bmatrix}\begin{Bmatrix} u_i \\ v_i \\ u_j \\ v_j \end{Bmatrix}
$$

Load and boundary conditions (BC):

$$
u_1 = v_1 = u_3 = v_3 = 0, \qquad F_{2X} = P_1, \; F_{2Y} = P_2
$$

$$
\sigma_1 = \frac{E}{L}\frac{\sqrt{2}}{2}\begin{bmatrix} -1 & -1 & 1 & 1 \end{bmatrix}\frac{L}{EA}\begin{Bmatrix} 0 \\ 0 \\ P_1 \\ P_2 \end{Bmatrix} = \frac{\sqrt{2}}{2A}(P_1 + P_2)
$$

Condensed equation & solving

$$
\frac{EA}{2L}\begin{bmatrix} 2 & 0 \\ 0 & 2 \end{bmatrix}\begin{Bmatrix} u_2 \\ v_2 \end{Bmatrix} = \begin{Bmatrix} P_1 \\ P_2 \end{Bmatrix} \rightarrow \begin{Bmatrix} u_2 \\ v_2 \end{Bmatrix} = \frac{L}{EA}\begin{Bmatrix} P_1 \\ P_2 \end{Bmatrix}
$$

$$
\sigma_2 = \frac{E}{L}\frac{\sqrt{2}}{2}\begin{bmatrix} 1 & -1 & -1 & 1 \end{bmatrix}\frac{L}{EA}\begin{Bmatrix} P_1 \\ P_2 \\ 0 \\ 0 \end{Bmatrix} = \frac{\sqrt{2}}{2A}(P_1 - P_2)
$$

Assignement #1*

Start a Matlab code for solving Example2 from Scratch

A simple plane truss is made of two identical bars (with E, A, and L), and loaded as shown in the figure. Find

1) displacement of node 2;

2) stress in each bar.