CE394M: 1D-Finite Element Method

Krishna Kumar

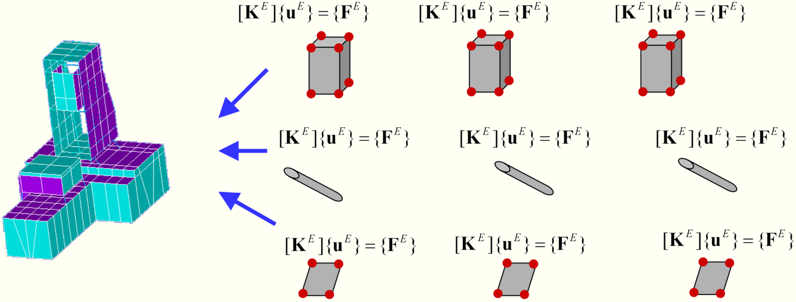
University of Texas at Austin

krishnak@utexas.edu

February 9, 2019

Overview

1. FEM workflow
2. 1D FEM
3. Assembly
4. Errors in the FEM
5. A case-study of a FE failure
6. 2D truss analysis

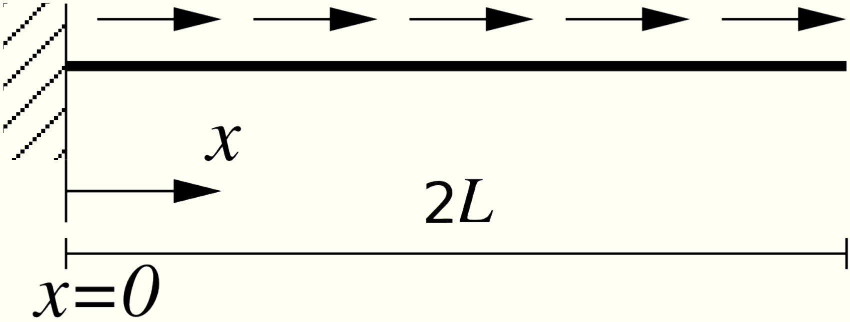


$$[\mathbf{K}]\{\mathbf{u}\} = \{\mathbf{F}\} \longrightarrow \{\mathbf{u}\} = [\mathbf{K}]^{-1}\{\mathbf{F}\}$$

Finite Element Analysis

FEM is a systematic procedure for approximating differential equations. For any problem in any spatial dimension it follows the same steps:

1D Finite Element Analysis of a cantilever beam



1D cantilever beam

Assume L as unit length $L = 1$. Unit force $f = 1$.

1D Finite Element Analysis of a cantilever beam

What element should be used?

Linear elements

Qudrilateral element

1D discretization of a cantilever beam

1D FEM: Shape functions and derivatives

Shape function **N**:

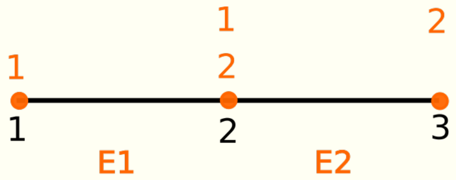
B is the derivatives of the shape functions:

In matrix format:

Element stiffness k_e:

Right-hand side vector b_e is:

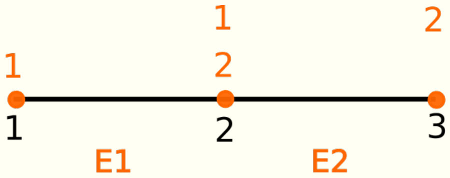
1D FEM: Assembly



Element stiffness k_e:

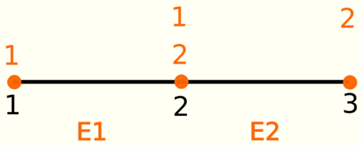
$$k_e = \begin{bmatrix} k_{11} & k_{12} \\ k_{21} & k_{22} \end{bmatrix}$$

$$= EA \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$



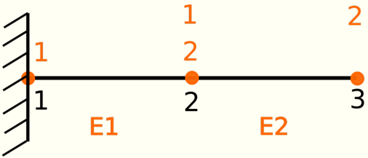
	element 1	element 2
Local dof		
Global dof		

1D FEM: Global stiffness matrix



Global stiffness $\mathbf{K}$:

Boundary conditions



Linear elements

Global stiffness **K**:

Applying boundary conditions: Approach I

Consider a global system that has already been assembled by adding the contribution of each element:

$$\begin{bmatrix} K_{11} & K_{12} & 0 & 0 & 0 \\ K_{21} & K_{22} & K_{23} & 0 & 0 \\ 0 & K_{32} & K_{33} & K_{34} & 0 \\ 0 & 0 & K_{43} & K_{44} & K_{45} \\ 0 & 0 & 0 & K_{54} & K_{55} \end{bmatrix} \begin{bmatrix} a_1 \\ a_2 \\ a_3 \\ a_4 \\ a_5 \end{bmatrix} = \begin{bmatrix} b_1 \\ b_2 \\ b_3 \\ b_4 \\ b_5 \end{bmatrix}$$

To apply a displacement of g at node one ($a_1 = g$):

Applying boundary conditions: Approach II

Prescribe the displacement at nodes 1 & 2, all entries in rows one and two in $\mathbf{K}$ and $\mathbf{b}$ will be equal to zero:

Since the g terms are known, we can take them to the right-hand side of the equation:

Global system of equations

Assemble the global system of equations:

$$\mathbf{K} = \begin{bmatrix} k_{11}^1 & k_{12}^1 & 0 \\ k_{21}^1 & k_{22}^1 + k_{11}^2 & k_{12}^2 \\ 0 & k_{21}^2 & k_{22}^2 \end{bmatrix} \begin{bmatrix} a_1 \\ a_2 \\ a_3 \end{bmatrix} = \begin{bmatrix} b_1 \\ b_2 \\ b_3 \end{bmatrix}$$

Applying boundary conditions

Assemble the global system of equations:

$$\begin{bmatrix} 1 & -1 & 0 \\ -1 & 2 & -1 \\ 0 & -1 & 1 \end{bmatrix} \begin{bmatrix} 0 \\ a_2 \\ a_3 \end{bmatrix} = \begin{bmatrix} 1/2 \\ 1 \\ 1/2 \end{bmatrix}$$

We need to apply the boundary condition $u(0) = 0$, which requires that $a_1 = 0$. The simplest way to impose this condition is to delete the first row and column of the stiffness matrix:

Solving this system we have:

Analytical solution of a 1D cantilever beam

The Euler–Bernoulli equation describes the relationship between the beam's deflection and the applied load:

$$-EA\frac{d^2u}{dx^2} = 1$$

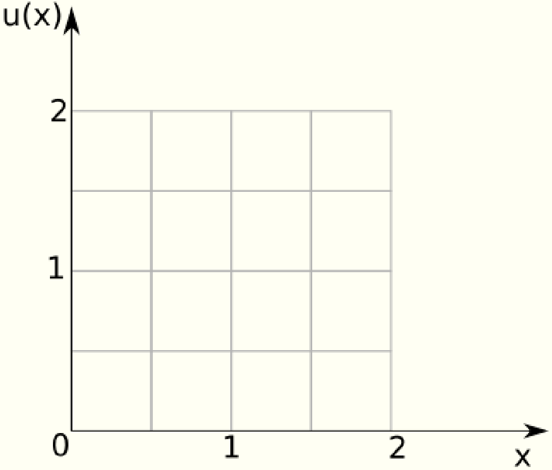
The exact solution is:

$$u = \frac{1}{EA}(\frac{-x^2}{2} + Cx + D)$$

Using the boundary conditions: $u(0) = 0$ and $\frac{du(2)}{dx} = 0$.

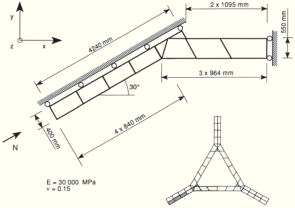
Error in the Finite Element Methods

This nodal '*exactness*' means that looking at error in the displacement at the nodes does not tell us about the error. Displacement error norm:



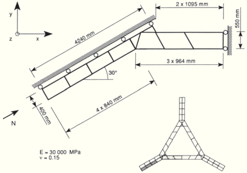
Sleipner A offshore platform sprang leak

Platform with a reinforced concrete base structure.



Concrete substructure during manufacturing on shore. Finite element mesh of a tricell detail. A tricell is a triangular concrete frame placed where three cells meet.

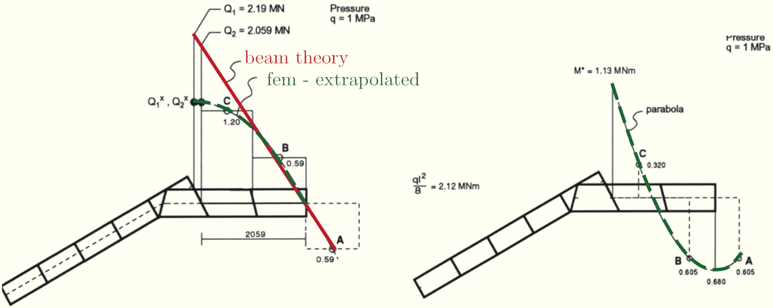
Platform with a reinforced concrete base structure.



Concrete substructure during manufacturing on shore. Finite element mesh of a tricell detail. A tricell is a triangular concrete frame placed where three cells meet.

Using the FE program NASTRAN, the shear stresses in the tricells were under-estimated by 47%. First, the chosen finite element mesh was exceedingly coarse so that the finite element shear stress was significantly too small.

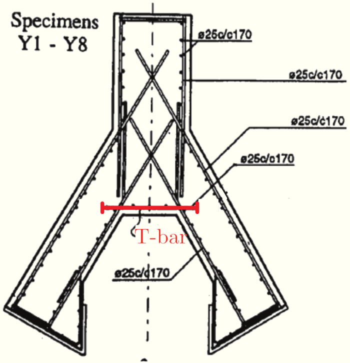
Sleipner A offshore platform sprang leak

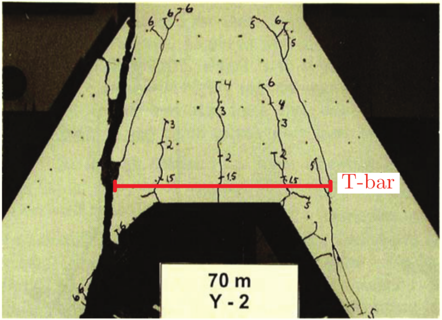


Comparison of the finite element shear forces (left) and moments (right) with the beam theory solution.

Second, the shear stresses at the boundary have been quadratically extrapolated using the shear forces at points A, B and C. We know however from beam theory that the shear force distribution is linear so that the shear force at the boundary is underestimated by $\approxeq 40\%$

Sleipner A offshore platform sprang leak



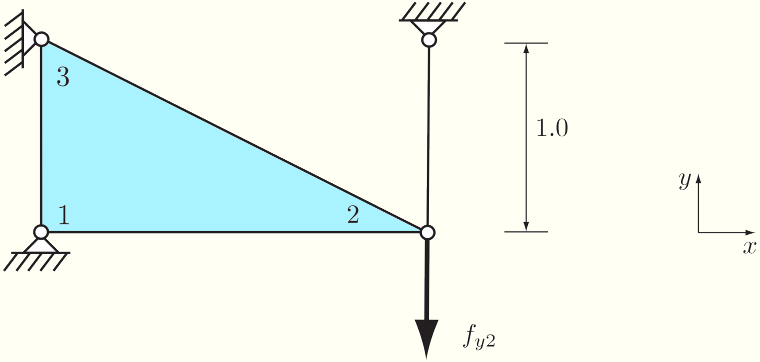


Experimental investigation of the tricell failure. As can be concluded from the failure mode the T-bar length is too short.

To make matters worse, the necessary reinforcement was automatically dimensioned based on the FE results without any checking by an engineer.

2D truss analysis

Consider the following structural system consisting of a three-noded triangle and a cable element (i.e. two-noded one-dimensional element):



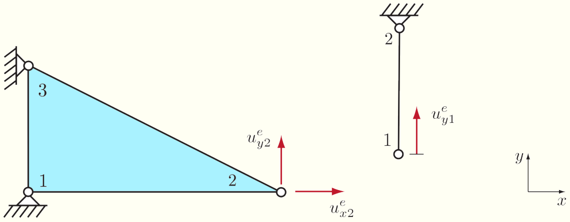
2D truss analysis

The triangle element is fixed at the (local) nodes 1 and 3 and its stiffness matrix for the unconstrained degrees of freedom at node 2 is

$$\mathbf{K} = 10^9 \begin{array}{c} \begin{bmatrix} 1.97 & 0 \\ 0 & 0.66 \end{bmatrix} \end{array} \begin{array}{l} u^e_{x2} \\ u^e_{y2} \end{array}$$

For the cable, the product of the Young's modulus and cross-sectional area is $EA = 1.0 \cdot 10^9$. Further, the system is loaded with a nodal force of $f_{y2} = -10000$.
Inserting EA and the length $h = 1.0$ gives the stiffness matrix for a two-noded one-dimensional element as:

To compose the global stiffness matrix we consider the correspondence between the element specific (local) degrees of freedom of the triangle and the cable



In the global stiffness matrix, the element stiffness matrix components corresponding to u^e_{y2} of the triangle and u^e_{y1} of the cable are add up:

2D truss analysis

Hence, the final equation system for determining the nodal displacements is: