# CE394M: 1D-Finite Element Method

Krishna Kumar

University of Texas at Austin

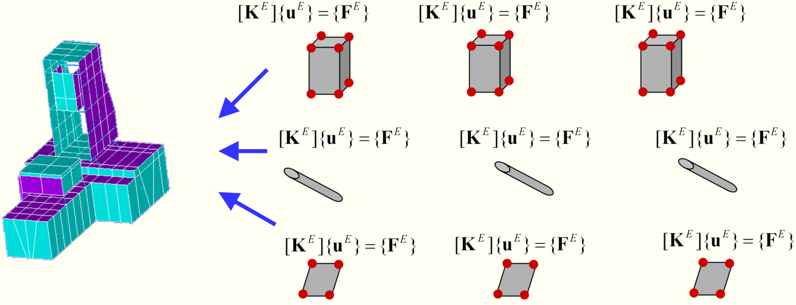
*krishnak@utexas.edu*

February 15, 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

# Finite Element Analysis



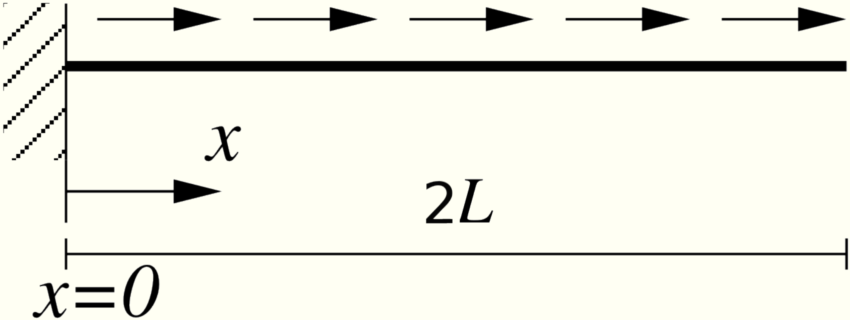

$$[\mathbf{K}]\{\mathbf{u}\} = \{\mathbf{F}\} \longrightarrow \{\mathbf{u}\} = [\mathbf{K}]^{-1}\{\mathbf{F}\}$$

# Finite Element Analysis

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

1. Identify the equation of interest
2. Cast the equation of interest in a weak form
3. Select a finite element type
4. Construct the element matrix and vector
5. Assemble the global matrix and vector and apply boundary conditions
6. Solve the system of linear equations

# 1D Finite Element Analysis of a cantilever beam

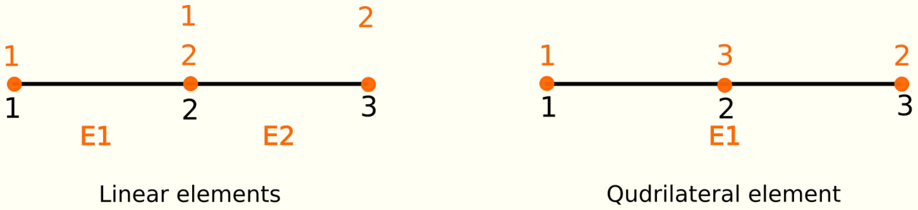



1D cantilever beam

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

# 1D Finite Element Analysis of a cantilever beam

What element should be used?




1D discretization of a cantilever beam

# 1D FEM: Shape functions and derivatives

Shape function **N**:

$$\mathbf{N} = \left[1 - \frac{x}{L} \qquad \frac{x}{L}\right]$$

$$\mathbf{N} = [1 - x \qquad x]$$

**B** is the derivatives of the shape functions:

$$\mathbf{B} = \left[\frac{dN_1(x)}{dx} \qquad \frac{dN_2(x)}{dx}\right]$$

$$\mathbf{B} = \left[\frac{-1}{L} \qquad \frac{1}{L}\right] = [-1 \quad 1]$$

In matrix format: $u_h = \mathbf{N}\mathbf{a_e}$ and $\epsilon_h = \mathbf{B}\mathbf{a_e}$.

# 1D FEM: Stiffness and force

Element stiffness $k_e$:

$$k_e = \int \mathbf{B}^T EA \, \mathbf{B} \, \mathrm{d}x = \int_0^L \begin{bmatrix} \frac{dN_1}{dx} \\ \frac{dN_2}{dx} \end{bmatrix} EA \begin{bmatrix} \frac{dN_1}{dx} & \frac{dN_2}{dx} \end{bmatrix} dx$$

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

Right-hand side vector $b_e$ is:

$$b_e = \int \mathbf{N}^T f \, \mathrm{d}x = \int_0^L \begin{bmatrix} -\frac{x}{L} + 1 \\ \frac{x}{L} \end{bmatrix} = \begin{bmatrix} -\frac{x^2}{2L} + x \\ \frac{x^2}{2L} \end{bmatrix}_0^L = \begin{bmatrix} -\frac{x^2}{2} + x \\ \frac{x^2}{2} \end{bmatrix}_0^1$$

$$b_e = \begin{bmatrix} \frac{1}{2} \\ \frac{1}{2} \end{bmatrix}$$

## 1D FEM: Stiffness and force

Element stiffness $k_e$:

$$k_e = \int \mathbf{B}^T EA \mathbf{B} \,\mathrm{d}x = \int_0^L \begin{bmatrix} \frac{dN_1}{dx} \\ \frac{dN_2}{dx} \end{bmatrix} EA \begin{bmatrix} \frac{dN_1}{dx} & \frac{dN_2}{dx} \end{bmatrix} dx$$

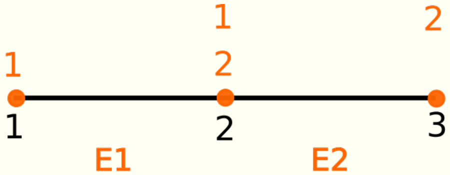
$$k_e = EA \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

Right-hand side vector $b_e$ is:

$$b_e = \int \mathbf{N}^T f \,\mathrm{d}x = \int_0^L \begin{bmatrix} -\frac{x}{L} + 1 \\ \frac{x}{L} \end{bmatrix} = \begin{bmatrix} -\frac{x^2}{2L} + x \\ \frac{x^2}{2L} \end{bmatrix}_0^L = \begin{bmatrix} -\frac{x^2}{2} + x \\ \frac{x^2}{2} \end{bmatrix}_0^1$$

$$b_e = \begin{bmatrix} \frac{1}{2} \\ \frac{1}{2} \end{bmatrix}$$

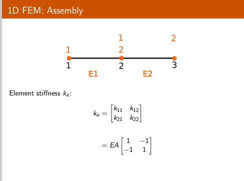
The force vector $b_e$ should be equal to the total sum of all the forces acting on the element. Here we have a unit force over a unit length so the total force is $1 \cdot 1 = 1$. This force of 1 is distributed over the two nodes as $\begin{bmatrix} \frac{1}{2} \\ \frac{1}{2} \end{bmatrix}$.




Element stiffness $k_e$:

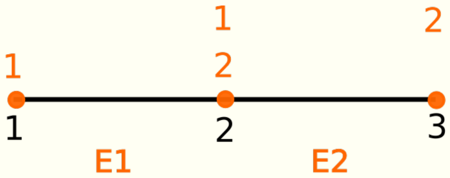
$$k_e = \begin{bmatrix} k_{11} & k_{12} \\ k_{21} & k_{22} \end{bmatrix}$$

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



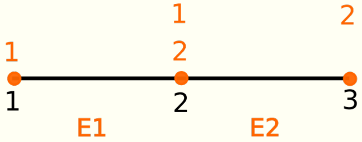

To perform a FE analysis, the element matrices and vectors are computed for each element and inserted into the global stiffness matrix **K** and global right-hand side vector **b**. The system $\mathrm{Ka} = \mathrm{b}$ is then solved to yield **a**. The process of inserting the element matrices and vectors into their global counterparts is known as assembly. A local (element) degree of freedom corresponds to a global degree of freedom, and an entry in a local matrix or vector is copied to its corresponding position in the global matrix.

# 1D FEM: Assembly




| | element 1 | | element 2 | |
|---|---|---|---|---|
| Local dof | 1 | 2 | 1 | 2 |
| Global dof | 1 | 2 | 2 | 3 |

# 1D FEM: Global stiffness matrix



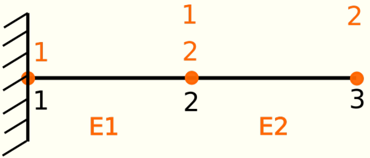

Global stiffness **K**:

$$k_e = \begin{bmatrix} k_{11}^1 & k_{12}^1 & 0 \\ k_{21}^1 & k_{22}^1 + k_{11}^2 & k_{12}^2 \\ 0 & k_{21}^2 & k_{22}^2 \end{bmatrix}$$

$$= EA \begin{bmatrix} 1 & -1 & 0 \\ -1 & 1+1 & -1 \\ 0 & -1 & 1 \end{bmatrix} = EA \begin{bmatrix} 1 & -1 & 0 \\ -1 & 2 & -1 \\ 0 & -1 & 1 \end{bmatrix}$$

Where superscripts in $k_{ij}^e$ denotes element numbering.

# Boundary conditions




Linear elements

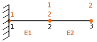
Global stiffness $\mathbf{K}$:

$$\mathbf{K} = \begin{bmatrix} k_{11}^1 & k_{12}^1 & 0 \\ k_{21}^1 & k_{22}^1 + k_{11}^2 & k_{12}^2 \\ 0 & k_{21}^2 & k_{22}^2 \end{bmatrix}$$

$$= EA \begin{bmatrix} 1 & -1 & 0 \\ -1 & 1 + 1 & -1 \\ 0 & -1 & 1 \end{bmatrix} = EA \begin{bmatrix} 1 & -1 & 0 \\ -1 & 2 & -1 \\ 0 & -1 & 1 \end{bmatrix}$$

Where superscripts in $k_{ij}^e$ denotes element numbering.

## Boundary conditions




Linear elements

Global stiffness $\mathbf{K}$:

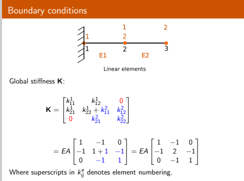
$$\mathbf{K} = \begin{bmatrix} k_{11}^1 & k_{12}^1 & 0 \\ k_{21}^1 & k_{22}^1 + k_{11}^2 & k_{12}^2 \\ 0 & k_{21}^2 & k_{22}^2 \end{bmatrix}$$

$$= EA \begin{bmatrix} 1 & -1 & 0 \\ -1 & 1+1 & -1 \\ 0 & -1 & 1 \end{bmatrix} = EA \begin{bmatrix} 1 & -1 & 0 \\ -1 & 2 & -1 \\ 0 & -1 & 1 \end{bmatrix}$$

Where superscripts in $k_{ij}^e$ denotes element numbering.

**Neumann (force) boundary conditions** appear naturally in the weak form of the equilibrium equation, and therefore do not require special consideration. In the context of variational methods, they are sometimes referred to as '*natural boundary conditions*'.

However, **Dirichlet (displacement) boundary conditions** still need to be applied. If we fail to apply proper displacement boundary conditions, the stiffness matrix will be singular, meaning that there is no unique solution. In the context of variational methods, displacement boundary conditions are often referred to as '*essential boundary conditions*'.




There are two approaches that are commonly used to apply displacement boundary conditions. One is to modify terms in the global stiffness matrix and place the displacement boundary condition in the right-hand side vector, and the other is to eliminate the degrees of freedom associated with a displacement boundary condition from the global system of equations.

# 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 ($a_1 = g$):
we can zero the first row of the matrix, place a 'one' on the diagonal and set $b_1 = g$,

$$\begin{bmatrix} 1 & 0 & 0 & 0 & 0 \\ K_{21} & K_{22} & K_{23} & 0 & 0 \\ 0 & K_{32} & K_{33} & K_{34} & 0 \\ 0 & 0 & K_{43} & K_{44} & K_{45} \\ 0 & 0 & 0 & K_{54} & K_{55} \end{bmatrix} \begin{bmatrix} a_1 \\ a_2 \\ a_3 \\ a_4 \\ a_5 \end{bmatrix} = \begin{bmatrix} g \\ b_2 \\ b_3 \\ b_4 \\ b_5 \end{bmatrix}$$

## 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 ($a_1 = g$):
we can zero the first row of the matrix, place a 'one' on the diagonal and set $b_1 = g$,

$$\begin{bmatrix} 1 & 0 & 0 & 0 & 0 \\ K_{21} & K_{22} & K_{23} & 0 & 0 \\ 0 & K_{32} & K_{33} & K_{34} & 0 \\ 0 & 0 & K_{43} & K_{44} & K_{45} \\ 0 & 0 & 0 & K_{54} & K_{55} \end{bmatrix} \begin{bmatrix} a_1 \\ a_2 \\ a_3 \\ a_4 \\ a_5 \end{bmatrix} = \begin{bmatrix} g \\ b_2 \\ b_3 \\ b_4 \\ b_5 \end{bmatrix}$$

Multiplying the top row of the matrix by the vector $\mathbf{a}$, we can see that solving this system of equations will lead to $a_1 = g$. There is however a significant drawback to this technique: if the matrix was originally symmetric, we have destroyed the symmetry. We will not to be able to use any specialised linear solvers that exploit symmetry of a matrix, which will possibly double the solution time. For $g = 0$ we could also zero the first column and preserve symmetry, but this is not possible for $g \neq 0$.

# Applying boundary conditions: Approach II

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

$$\begin{bmatrix} 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 \\ 0 & K_{32} & K_{33} & K_{34} & 0 \\ 0 & 0 & K_{43} & K_{44} & K_{45} \\ 0 & 0 & 0 & K_{54} & K_{55} \end{bmatrix} \begin{bmatrix} g_1 \\ g_2 \\ a_3 \\ a_4 \\ a_5 \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ b_3 \\ b_4 \\ b_5 \end{bmatrix}$$

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

$$\begin{bmatrix} K_{33} & K_{34} & 0 \\ K_{43} & K_{44} & K_{45} \\ 0 & K_{54} & K_{55} \end{bmatrix} \begin{bmatrix} a_3 \\ a_4 \\ a_5 \end{bmatrix} = \begin{bmatrix} b_3 \\ b_4 \\ b_5 \end{bmatrix} - \begin{bmatrix} 0 & K_{32} \\ 0 & 0 \\ 0 & 0 \end{bmatrix} \begin{bmatrix} g_1 \\ g_2 \end{bmatrix}$$

## Applying boundary conditions: Approach II

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

$$\begin{bmatrix} 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 \\ 0 & K_{32} & K_{33} & K_{34} & 0 \\ 0 & 0 & K_{43} & K_{44} & K_{45} \\ 0 & 0 & 0 & K_{54} & K_{55} \end{bmatrix} \begin{bmatrix} g_1 \\ g_2 \\ a_3 \\ a_4 \\ a_5 \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ b_3 \\ b_4 \\ b_5 \end{bmatrix}$$

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

$$\begin{bmatrix} K_{33} & K_{34} & 0 \\ K_{43} & K_{44} & K_{45} \\ 0 & K_{54} & K_{55} \end{bmatrix} \begin{bmatrix} a_3 \\ a_4 \\ a_5 \end{bmatrix} = \begin{bmatrix} b_3 \\ b_4 \\ b_5 \end{bmatrix} - \begin{bmatrix} 0 & K_{32} \\ 0 & 0 \\ 0 & 0 \end{bmatrix} \begin{bmatrix} g_1 \\ g_2 \end{bmatrix}$$

The second approach is to eliminate the degrees of freedom where the displacement is prescribed. Firstly, recall that wherever a displacement condition is applied, the weight function must be equal to zero.

Now we have smaller matrix and any symmetry is preserved. This technique can be applied element-wise during assembly of the global matrix and it is used by finite element programs aimed at solid and structural mechanics. It is however more complicated to program than the first approach.

# Global system of equations

Assemble the global system of equations:

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

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

# 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:

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

Solving this system we have: $a^T = (1/EA)\,[0 \quad 1.5 \quad 2.0]$.

# 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 $\frac{du(2)}{dx} = 0$.

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

## 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 $\frac{du(2)}{dx} = 0$.

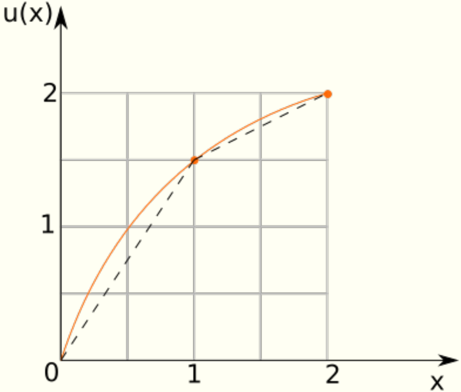
$$u = \frac{1}{EA}(\frac{-x^2}{2} + 2)$$

The finite element solution is exact at the nodes, but it is not exact between the nodes. This '*exactness*' is a feature of finite element methods in one dimension, but it does not carry over to higher dimensions.

# 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:

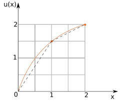
$$e_u = \left( \int_0^L (u - u_h)^2 \, \mathrm{d}x \right)$$



## 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:

$$e_u = \left( \int_0^L (u - u_h)^2 \, \mathrm{d}x \right)$$

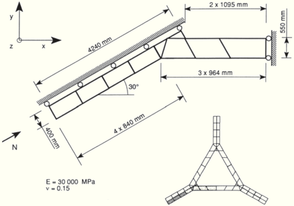



The finite element solution is exact at the nodes, but it is not exact between the nodes. This '*exactness*' is a feature of finite element methods in one dimension, but it does not carry over to higher dimensions.

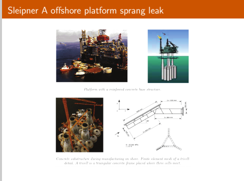
Since the exact solution is quadratic, the finite element solution would be exact if you use elements which are quadratic or higher.

# Sleipner A offshore platform sprang leak


*Platform with a reinforced concrete base structure.*

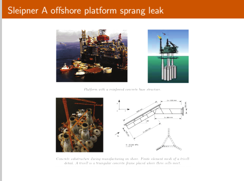

*Concrete substructure during manufacturing on shore. Finite element mesh of a tricell detail. A tricell is a triangular concrete frame placed where three cells meet.*




The Sleipner A platform produces oil and gas in the North Sea and is supported on the seabed at a water depth of 82m. It is a platform with a concrete gravity base structure consisting of 24 cells. Four cells are elongated to shafts supporting the platform deck.

The failure involved a total economic loss of about 700 million dollars.

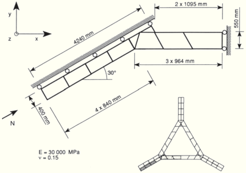
All 14 people onboard the platform was rescued by nearby boats without injuries.




The first concrete base structure for Sleipner A offshore platform sprang leak and sank on 23 August 1991 during the controlled sinking operation in the open sea. The concrete base structure was constructed on land, pulled out to open sea and the cells were in the process being flooded with water when the accident occurred. During this operation the water level in the cells is higher than the sea level which leads to substantial pressure loading on the cell walls. The conclusion of the subsequent investigation was that the loss was caused by a failure in a cell wall connection due to shear failure, resulting in a serious crack and leakage. The wall failed as a result of a combination of errors in the finite element analysis, the interpretation of the finite element results and insufficient anchorage of the shear reinforcement in the tricells.

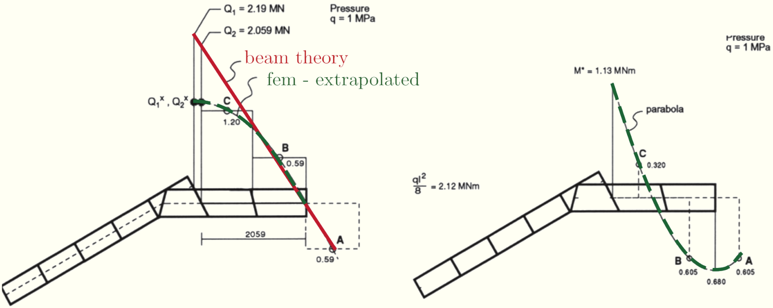
# Sleipner A offshore platform sprang leak

*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.*

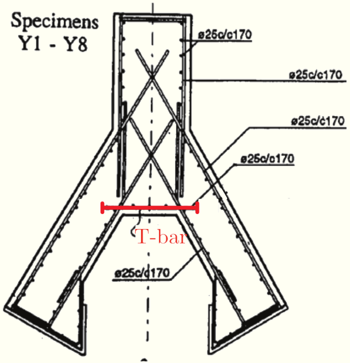
Using the FE program NASTRAN, the shear stresses in the tricells were under-estimated by 47%. First, the chosen finite element mesh was exceedingly coarse so that the finite element shear stress was significantly too small.

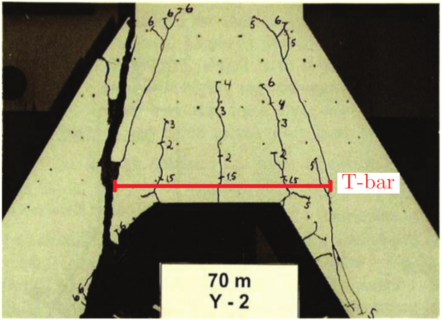



*Comparison of the finite element shear forces (left) and moments (right) with the beam theory solution.*

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



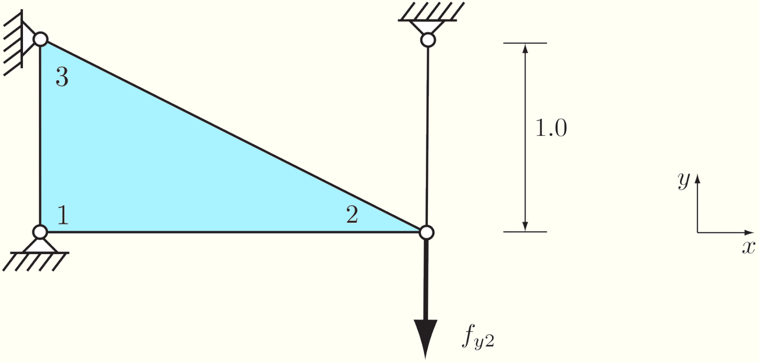




*Experimental investigation of the tricell failure. As can be concluded from the failure mode the T-bar length is too short.*

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):



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

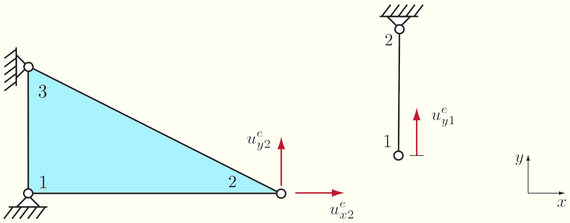
$$\mathbf{K} = 10^9 \begin{array}{cc} \begin{bmatrix} 1.97 & 0 \\ 0 & 0.66 \end{bmatrix} & \begin{array}{c} u^e_{x2} \\ u^e_{y2} \end{array} \end{array}$$

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:

$$\mathbf{K}^e = \frac{EA}{h} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = 10^9 \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

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:

$$\mathbf{K}^e = 10^9 \begin{bmatrix} 1.97 & 0 \\ 0 & 1 + 0.66 \end{bmatrix}$$

# 2D truss analysis

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

$$10^9 \begin{bmatrix} 1.97 & 0 \\ 0 & 1.66 \end{bmatrix} \begin{bmatrix} u_x \\ u_y \end{bmatrix} = \begin{bmatrix} 0 \\ -1000 \end{bmatrix} \qquad (\mathbf{Ka} = \mathbf{f})$$

$$\begin{bmatrix} u_x \\ u_y \end{bmatrix} = \begin{bmatrix} 0 \\ 6.02 \times 10^{-06} \end{bmatrix}$$