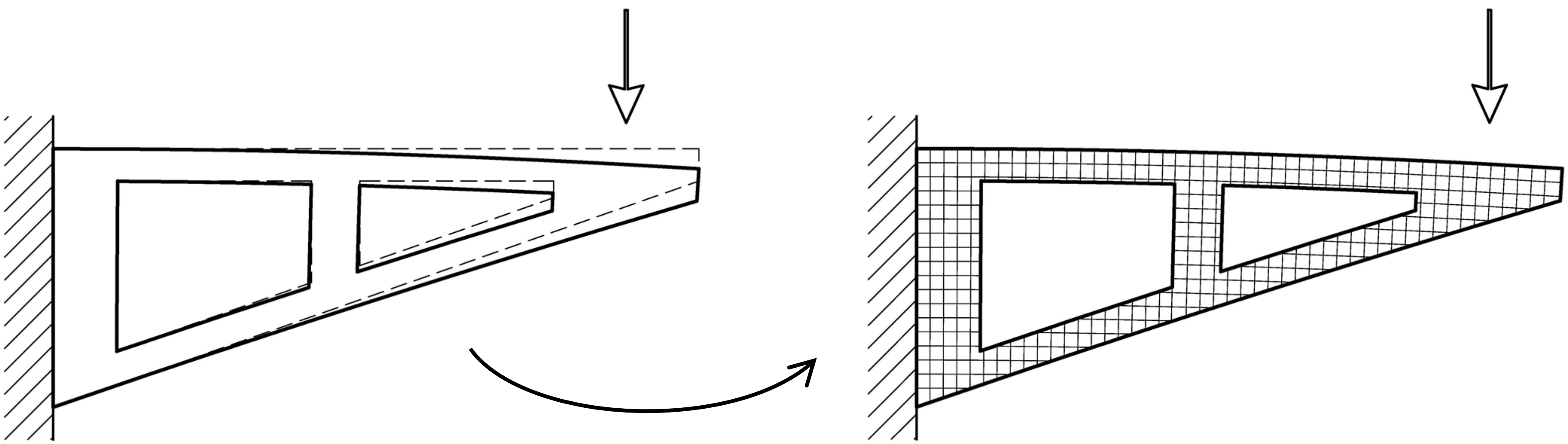
MAE 656 - Advanced Computer Aided Design

01. Introduction – Doc 05

Numerical Simulation
2D Solid Structures

Introduction

As we did with the bar elements (beams and trusses) we want to find a relation between the displacements and the forces in the structure:



If we cannot solve the continuum, we can try to find the solution for a finite number of points!

Introduction

To do so, we need an equation capable of relating the forces and the displacements of a solid structure.

PRINCIPLE OF VIRTUAL WORK:

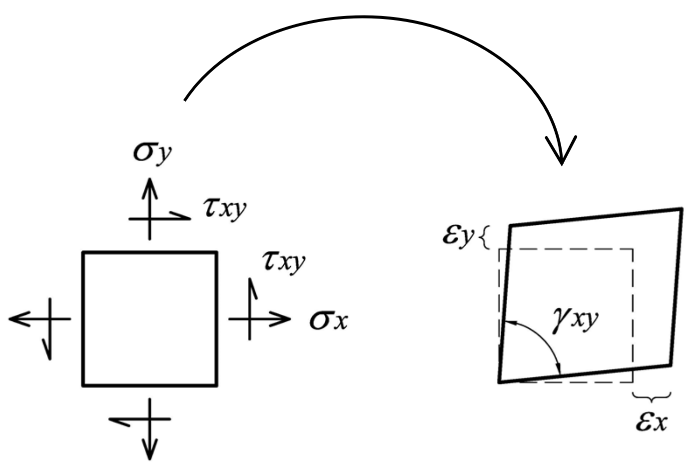
$$\int_V \delta\varepsilon^T \cdot \sigma \cdot dV = \int_V \delta u^T \cdot b \cdot dV + \int_A \delta u^T \cdot q \cdot dA + \sum \delta u^T{}_i \cdot P_i$$

If, instead of solving a structural problem, we want to solve another type of problem (i.e. fluid), we will need to use a different equation (for the fluid case, Navier-Stokes)

Introduction

Brief note on the Theory of Elasticity:

We will work with elastic materials. This means that there is a linear relation between the strains and stresses of the material:



$$\sigma = D \cdot \varepsilon$$

$$D = \frac{1}{1 - \nu_{xy} \cdot \nu_{yx}} \cdot \begin{bmatrix} E_x & \nu_{xy} \cdot E_x & 0 \\ \nu_{yx} \cdot E_y & E_y & 0 \\ 0 & 0 & (1 - \nu_{xy} \cdot \nu_{yx}) \cdot G_{xy} \end{bmatrix}$$

Introduction

D is the stiffness matrix of the material, which is defined as:

$$D = \frac{1}{1 - \nu_{xy} \cdot \nu_{yx}} \cdot \begin{bmatrix} E_x & \nu_{xy} \cdot E_x & 0 \\ \nu_{yx} \cdot E_y & E_y & 0 \\ 0 & 0 & (1 - \nu_{xy} \cdot \nu_{yx}) \cdot G_{xy} \end{bmatrix}$$

D is always symmetric, which means:

$$\nu_{xy} \cdot E_x = \nu_{yx} \cdot E_y$$

And, if the material is isotropic:

$$E_x = E_y = E$$

$$\nu_{xy} = \nu_{yx} = \nu$$

2D FEM Formulation

We will develop the formulation for the 2D plane stress problem. It is enough for our purpose, knowing how a FEM code works, and the formulation is easier to follow.

In the 2D plane stress problem, the PVW can be rewritten as:

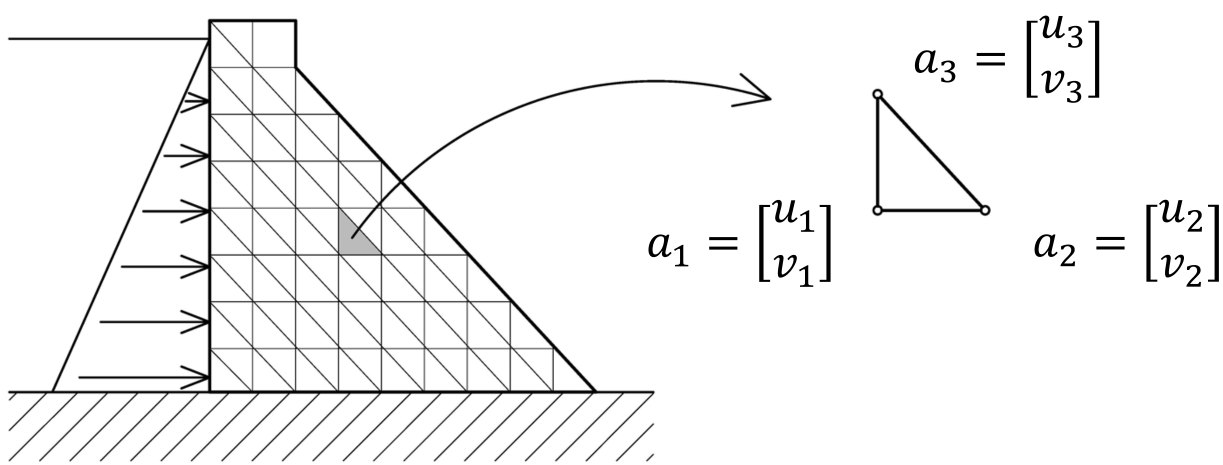
$$\int_V \delta\varepsilon^T \cdot \sigma \cdot dV = \int_V \delta u^T \cdot b \cdot dV + \int_A \delta u^T \cdot q \cdot dA + \sum \delta u^T{}_i \cdot P_i$$

$$\int_A \delta\varepsilon^T \cdot \sigma \cdot t \cdot dA = \int_A \delta u^T \cdot b \cdot t \cdot dA + \int_l \delta u^T \cdot q \cdot t \cdot dl + \sum \delta u^T{}_i \cdot P_i$$

2D FEM Formulation

The FEM method discretizes the solid in a finite number of elements.

And, the displacement field in those elements is discretized by the displacement of a finite number of nodes



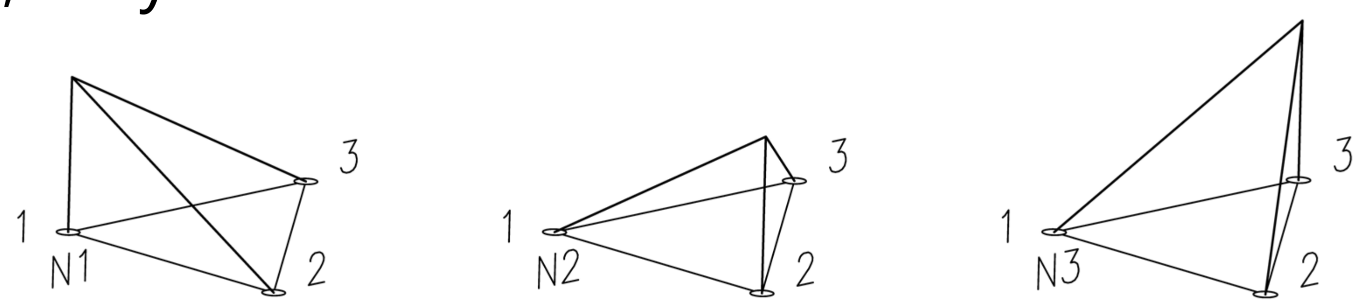
2D FEM Formulation

If we get to know the displacements of the three nodes of the element, it is possible to calculate the displacement of any point of the element with a linear combination of these three displacements:

$$u = N_1 \cdot u_1 + N_2 \cdot u_2 + N_3 \cdot u_3$$

$$v = N_1 \cdot v_1 + N_2 \cdot v_2 + N_3 \cdot v_3$$

Ni are the shape functions and, for a linear triangular element, they look like this:



2D FEM Formulation

and their expression is:

$$u = N_i = \frac{1}{2A}(a_i + b_i \cdot x + c_i \cdot y)$$

with:

$$a_i = x_j \cdot y_k - x_k \cdot y_j$$

$$b_i = y_j - y_k$$

$$c_i = x_k - x_j$$

Being x_i and y_i the coordinates of the element node i, and A the element area.

2D FEM Formulation

Before returning to the PVW, we will express the element strains as a function of the point displacement.

with:

$$\varepsilon_x = \frac{\partial u}{\partial x} = \frac{\partial N_1}{\partial x} u_1 + \frac{\partial N_2}{\partial x} u_2 + \frac{\partial N_3}{\partial x} u_3$$

$$\varepsilon_y = \frac{\partial v}{\partial y} = \frac{\partial N_1}{\partial y} v_1 + \frac{\partial N_2}{\partial y} v_2 + \frac{\partial N_3}{\partial y} v_3$$

$$\gamma_{xy} = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} = \frac{\partial N_1}{\partial y} u_1 + \frac{\partial N_1}{\partial x} v_1 + \frac{\partial N_2}{\partial y} u_2 + \frac{\partial N_2}{\partial x} v_2 + \frac{\partial N_3}{\partial y} u_3 + \frac{\partial N_3}{\partial x} v_3$$

This can be written in matrix form as:

2D FEM Formulation

Before returning to the PVW, we will express the element strains as a function of the point displacement.

$$\varepsilon = \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{bmatrix} = \begin{bmatrix} \frac{\partial N_1}{\partial x} & 0 & \frac{\partial N_2}{\partial x} & 0 & \frac{\partial N_3}{\partial x} & 0 \\ 0 & \frac{\partial N_1}{\partial y} & 0 & \frac{\partial N_2}{\partial y} & 0 & \frac{\partial N_3}{\partial y} \\ \frac{\partial N_1}{\partial y} & \frac{\partial N_1}{\partial x} & \frac{\partial N_2}{\partial y} & \frac{\partial N_2}{\partial x} & \frac{\partial N_3}{\partial y} & \frac{\partial N_3}{\partial x} \end{bmatrix} \cdot \begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{bmatrix}$$

This is **B** = ELEMENT DEFORMATION MATRIX

Therefore,

$$\varepsilon = B \cdot a$$

2D FEM Formulation

Using the shape functions defined previously, the element deformation matrix becomes:

$$B = \begin{bmatrix} b_1 & 0 & b_2 & 0 & b_3 & 0 \\ 0 & c_1 & 0 & c_2 & 0 & c_3 \\ c_1 & b_1 & c_2 & b_2 & c_3 & b_3 \end{bmatrix}$$

Note that B is divided in 3 matrices 3x2:

- The number of rows correspond to the number of strain values (ε_x, ε_y and γ_{xy})
- The number of columns correspond to the dimension of the space (2D), as we are integrating in x and y
- Finally, the number of matrices correspond to the number of points in which we discretize the Finite Element.

2D FEM Formulation

Let's return to the PVW:

$$\int_A \delta\varepsilon^T \cdot \sigma \cdot t \cdot dA = \int_A \delta u^T \cdot b \cdot t \cdot dA + \int_l \delta u^T \cdot q \cdot t \cdot dl + \sum \delta u^T{}_i \cdot P_i$$

As the integrals are additive, we can apply it to a single element and, afterwards, sum all elements. Therefore,

$$\int_{A_e} \delta\varepsilon^T \cdot \sigma \cdot t \cdot dA = \int_{A_e} \delta u^T \cdot b \cdot t \cdot dA + \int_{l_e} \delta u^T \cdot q \cdot t \cdot dl + \sum \delta u^T{}_i \cdot P_i$$

where,

$$\delta\varepsilon^T = \delta a^T \cdot B^T$$

$$\delta u^T = \delta a^T \cdot N^T$$

with, a, the displacements of the nodes of the element

2D FEM Formulation

So, the PVW can be rewritten as:

$$\int_{A_e} \delta a^T \cdot B^T \cdot \sigma \cdot t \cdot dA = \int_{A_e} \delta a^T \cdot N^T \cdot b \cdot t \cdot dA + \int_{l_e} \delta a^T \cdot N^T \cdot q \cdot t \cdot dl + \sum \delta a^T{}_i \cdot P_i$$

As the a values are constants, they can be taken out of the integrals:

$$\delta a^T \int_{A_e} B^T \cdot \sigma \cdot t \cdot dA = \delta a^T \cdot \left(\int_{A_e} N^T \cdot b \cdot t \cdot dA + \int_{l_e} N^T \cdot q \cdot t \cdot dl \right) + \delta a^T{}_i \sum P_i$$

And, as this expression has to be fulfilled for any possible virtual displacement δa. It can be removed from the equation.

2D FEM Formulation

$$\int_{A_e} B^T \cdot \sigma \cdot t \cdot dA = \left(\int_{A_e} N^T \cdot b \cdot t \cdot dA + \int_{l_e} N^T \cdot q \cdot t \cdot dl \right) + P_i$$

The stresses can be replaced by:

$$\sigma = D \cdot \varepsilon = D \cdot B \cdot a$$

Therefore,

$$\int_{A_e} B^T \cdot D \cdot B \cdot a \ \cdot t \cdot dA = \left(\int_{A_e} N^T \cdot b \cdot t \cdot dA + \int_{l_e} N^T \cdot q \cdot t \cdot dl \right) + P_i$$

$$\int_{A_e} B^T \cdot D \cdot B \cdot t \cdot dA \cdot a = \left(\int_{A_e} N^T \cdot b \cdot t \cdot dA + \int_{l_e} N^T \cdot q \cdot t \cdot dl \right) + P_i$$

2D FEM Formulation

Defining,

$$K_e = \int_{A_e} B^T \cdot D \cdot B \cdot t \cdot dA$$

as the element stiffness matrix. And,

$$P_{eq} = \left(\int_{A_e} N^T \cdot b \cdot t \cdot dA + \int_{l_e} N^T \cdot q \cdot t \cdot dl \right)$$

as the vector of equivalent forces. We finally obtain:

$$K_e \cdot a_e = P_e + P_i$$

2D FEM Formulation

The final expression of the stiffness matrix is:

$$K_e = \int_{A_e} B^T \cdot D \cdot B \cdot t \cdot dA$$

$$K_e = \int_{A_e} \begin{bmatrix} {B^T}_1 \\ {B^T}_2 \\ {B^T}_3 \end{bmatrix} \cdot D \cdot [B_1 \quad B_2 \quad B_3] \cdot t \cdot dA \qquad \text{(6x6)}$$

that can be divided in:

$$K_{ij} = \int_{A_e} {B^T}_i \cdot D \cdot B_j \cdot t \cdot dA \qquad \text{(2x2)}$$

2D FEM Formulation

$$K_{ij} = \int_{A_e} \frac{1}{2A_e} \begin{bmatrix} b_i & 0 & c_i \\ 0 & c_i & b_i \end{bmatrix} \cdot D \cdot \frac{1}{2A_e} \begin{bmatrix} b_j & 0 \\ 0 & c_j \\ c_j & b_j \end{bmatrix} \cdot t \cdot dA$$

As all terms are constant, they can be taken out of the integral. Obtaining, as final expression of the elemental stiffness matrix:

$$K_{ij} = \frac{t}{4A_e} \begin{bmatrix} b_i & 0 & c_i \\ 0 & c_i & b_i \end{bmatrix} \cdot D \cdot \begin{bmatrix} b_j & 0 \\ 0 & c_j \\ c_j & b_j \end{bmatrix} \cdot t$$

2D FEM Formulation

Once having calculated all stiffness matrices, they can be assembled into the structure stiffness matrix as:

$$\begin{bmatrix} \vdots \\ \vec{f_i} \\ \vdots \\ \vec{f_j} \\ \vdots \\ \vec{f_k} \end{bmatrix} = \begin{bmatrix} \cdots & \vdots & & \vdots & & \vdots \\ \cdots & k_{ii}{}^{e} & \cdots & k_{ij}{}^{e} & \cdots & k_{ik}{}^{e} \\ & \vdots & & \vdots & & \vdots \\ & k_{ji}{}^{e} & \cdots & k_{jj}{}^{e} & \cdots & k_{jk}{}^{e} \\ & \vdots & & \vdots & & \vdots \\ \cdots & k_{ki}{}^{e} & \cdots & k_{kj}{}^{e} & \cdots & k_{kk}{}^{e} \end{bmatrix} \begin{bmatrix} \vdots \\ \vec{a_i} \\ \vdots \\ \vec{a_j} \\ \vdots \\ \vec{a_k} \end{bmatrix}$$

→ Row i

→ Row j

→ Row k

→ Column k

→ Column j

→ Column i

Now, each k_{ij} is a 2x2 matrix and we have a total of 9 Kij matrices!

Implementation (1/2)

Having defined all elements required to solve a 2D plane stress structure with the FEM. The implementation in a code should be as follows:

1. Define the structure stiffness matrix.
   Its size is = (2 x # nodes) x (2 x # nodes)

2. For each material, compute the material stiffness matrix (D)

3. For each element in the structure, compute the element deformation matrix (B)

4. Compute the element stiffness matrix (with B and D)

5. Assemble the element stiffness matrix in the struct. stiffn. matrix

Implementation (1/2)

6. Define the load vector and the displacement vector
7. Remove all rows and columns of the elements with displacement equal to 0.0
8. Solve the linear system of equations to obtain the displacements of the structure
9. Calculate the reactions of the structure
10. Once the displacements of all structure nodes are known, it is possible to calculate the strains and stresses of each node using the element deformation matrix (B)

Conclusion

In order to solve the problem we need to know:

- Material properties: Ex, Ey, νxy, νyx and G (with these we can calculate D)
- Element coordinates and connectivities (to calculate B and to assemble the element). These is: solid geometry and FE mesh.
- Boundary conditions: displacements and loads applied on nodes

Once knowing all these parameters, and having introduced them in a FEM code, it is possible to perform a successful calculation!

Conclusion

It is important to know some basics of the formulation required by each element. This will tell us what parameters are required by the code:

Truss elements: To calculate the stiffness matrix the only parameter necessary is the Area of the element

Beam elements: These elements require knowing their area, and also their bending stiffness (inertia)

Solid Elements: These relate the strains and stresses with the material stiffness matrix. This matrix is function of the elastic parameters of the material. (a plane stress simulation also requires the element thickness)