Finite Element Analysis (CE5001) -Term Project

Balaji Sai Kumar Bandaru [101902003]
Indian Institute of Technology, Palakkad

July 8, 2020

Contents

Chapter 1

Introduction to Finite Element Analysis

In finite element method, the idea is to solve the mathematical model of the physical problem. The mathematical model is then simplified using assumptions. This will increase the error but decreases the computation power required and also decreases the complexity. One of basic assumption in all the basic finite element formulation is that material is isotropic and homogeneous. This makes it easier to use the constitutive relationship between strain and stress.

1.1 Steady State condition

The basic assumption while solving the problem is it is a steady state problem. In steady state condition, $\frac{\partial\{\}}{\partial t} = 0$. That means the primal field, (i.e u(x,y)-displacement and σ) are not time dependent.

1.2 Boundary Conditions

Usually, to solve a differential equation, boundary conditions should be specified. There are two types of boundary conditions

- Dirichlet boundary condition
- Neumann boundary condition

1.2.1 Dirichlet boundary condition

These boundary conditions are defined on primal fields. For 1D Elasticity problems, these are defined on u(x) where u is the displacement. For 1D Heat transfer problem, these are defined on temperature

1.2.2 Neumann boundary condition

These boundary conditions are defined on the derivative of the primal fields. For 1D Elasticity problems, these are defined on spacial derivative of displacement $\frac{du}{dx}$ and displacement. For 1D Heat transfer problem, these are defined on heat flux

1.3 Weak and Strong form of a differential equations

Strong form of a differential equation is the equation itself. It can be considered a strong form as it has the solution has to be of the order of the differential equation.
Conversion from strong form to weak form can be done using two different methods:

- Modified Galerkin Method
- Variational Method

Weak form of the differential equation becomes the basis for solving finite element method. After the strong form of the differential equation is converted to weak form, the domain is mapped to another domain in order to make the calculations easier. A mapping to bi-unit domain or Isoparametric mapping is used. Let Ω be mapped to ς domain whose limits are (-1,1). This mapping can be done using shape functions or basis functions. These shape functions are used to convert from Ω to ς domain.
Let the shape functions be N_i.
Then,

$$u = \sum N(\varsigma)_i u_i \tag{1.1}$$

For a 2D problem, it can be defined as:-

$$u = [N_1, N_2] \begin{bmatrix} u_1 \\ u_2 \end{bmatrix} \tag{1.2}$$

where N_1 and N_2 are $N_1(x, y)$ $N_2(x, y)$

Chapter 2

Theory

2.1 Constitutive Relationship

These are a sort of mechanical equation of state, and describe how the material is constituted mechanically. To solve all the equations we obtain, constitutive relations are used. These are dependent to material properties and loading conditions. The constitutive relation that is used for elasticity problems is

$$\epsilon_{ij} = D_{ijkl}\sigma_{kl} \tag{2.1}$$

After taking the inverse of $[D]$, we obtain the basic relation to solve all the elasticity problems. D is the complaince matrix.

$$\bar{\bar{D}} = \bar{\bar{C}}^{-1} \tag{2.2}$$

As D is a symmetric matrix, we obtain the following matrix.

$$\begin{bmatrix} \epsilon_x \\ \epsilon_y \\ \epsilon_z \\ \epsilon_{xy} \\ \epsilon_{yz} \\ \epsilon_{xz} \end{bmatrix} = \begin{bmatrix} \frac{1}{E} & \frac{-\mu}{E} & \frac{-\mu}{E} & 0 & 0 & 0 \\ \frac{-\mu}{E} & \frac{1}{E} & \frac{-\mu}{E} & 0 & 0 & 0 \\ \frac{-\mu}{E} & \frac{-\mu}{E} & \frac{-1}{E} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{G} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{G} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{G} \end{bmatrix} \begin{bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \sigma_{xy} \\ \sigma_{yz} \\ \sigma_{xz} \end{bmatrix}$$

where:

$$G = \frac{E}{2(1+\mu)}$$

$$\bar{\sigma} = \bar{\bar{D}}^{-1}\bar{\epsilon} \tag{2.3}$$

For a 2D problem, the constitutive relationship can be modified based on two conditions

- Plane strain
- Plane stress

2.2 Formulation of stiffness matrix

Formulation of stiffness matrix for an element can be done by the following equations. These equations are valid for obtaining stiffness matrix for 2D structure.

$$[k] = \int_A hBCB' dA$$

B, C and h are constants and hence can be removed from the integration.

$$[k] = hBCB' \int_A dA$$
$$[k] = hABCB' \tag{2.4}$$

where B can be obtained from equation 2.16
C can be obtained from equation 2.1
h is the thickness of the structure.
A is the area of the element

2.3 Formulation of Force vector

For the problem, there are two force vectors considered.
{F}- Nodal force vector
{q}- Equivalent Nodal force vector
Load potential V due to nodal load for 2D problem is given below:

$$V_N = F_{1,x}u_1 + F_{1,y}v_1 + F_{2,x}u_2 + F_{2,y}v_2 + F_{3,x}u_3 + F_{3,y}v_3 \tag{2.5}$$

Below figure 2.1 shows the nodal forces and the triangular element.

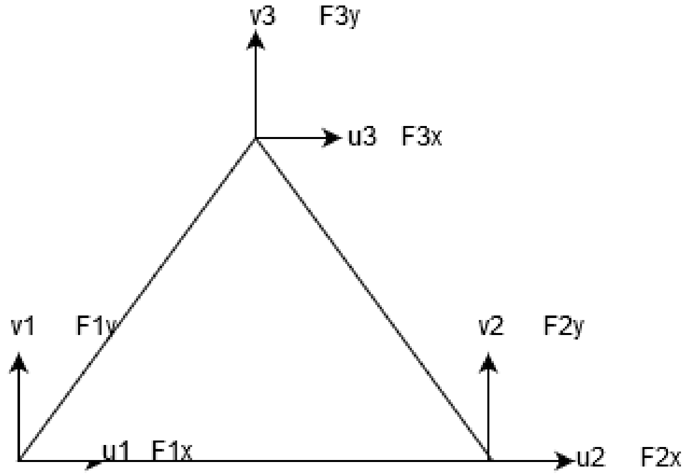

Figure 2.1: Nodal force on a triangular element

$$V_N = [u_1, v_1, u_2, v_2, u_3, v_3] \begin{bmatrix} F_{1,x} \\ F_{1,y} \\ F_{2,x} \\ F_{2,y} \\ F_{3,x} \\ F_{3,y} \end{bmatrix} \tag{2.6}$$

$$V_N = \{d\}'\{F\} \tag{2.7}$$

Equivalent nodal load vector for 2D problem is given below:

$$V_T = \{d\}'h \int N(s)T(s)det(J)ds \tag{2.8}$$

2.4 Global and local coordinate system

$$V_T = \{d\}[q] \tag{2.9}$$

The obtained stiffness matrix is of the order 20X20.

$$[k] = \begin{bmatrix} \alpha_{1,1} & \alpha_{1,2} & . & . & . & \alpha_{1,20} \\ \alpha_{1,1} & \alpha_{1,2} & . & . & . & \alpha_{2,20} \\ \alpha_{1,1} & \alpha_{1,2} & . & . & . & \alpha_{3,20} \\ . & . & . & . & . & . \\ . & . & . & . & . & . \\ \alpha_{19,1} & \alpha_{91,2} & . & . & . & \alpha_{19,20} \\ \alpha_{20,1} & \alpha_{20,2} & . & . & . & \alpha_{20,20} \end{bmatrix} \tag{2.10}$$

where $\alpha_{i,j}$ are obtained from corresponding elemental stiffness matrix. Elemental stiffness matrix are necessary to obtain the stresses and strains of the element.

2.5 Plane Stress vs Plane strain Condition

Plane stress and Plane strain conditions depend upon the dimensions of the structure for approximation in calculations. The assumptions in plane stress condition is given in 2.11

$$\sigma_{zz} = 0 \tag{2.11}$$

The assumptions in plane stress condition is given in 2.12

$$\epsilon_{zz} = 0 \tag{2.12}$$

When one dimension of the structure is much greater than other two dimensions, then the structure can be approximated into 2D. The constitutive relation is modified after introduction of these conditions. For plane stress condition,

$$\{\sigma\} = \begin{bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \tau_{xy} \end{bmatrix} = \frac{E}{(1-\mu^2)} \begin{bmatrix} 1 & \mu & 0 \\ -\mu & 1 & 0 \\ 0 & 0 & \frac{1-\mu}{2} \end{bmatrix} \begin{bmatrix} \epsilon_x \\ \epsilon_y \\ \epsilon_{xy} \end{bmatrix} \tag{2.13}$$

for plane strain conditions,

$$\{\sigma\} = \begin{bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \tau_{xy} \end{bmatrix} = \frac{E}{(1-2\mu)(1+\mu)} \begin{bmatrix} \mu & 1-\mu & 0 \\ 1-\mu & \mu & 0 \\ 0 & 0 & \frac{1-2\mu}{2} \end{bmatrix} \begin{bmatrix} \epsilon_x \\ \epsilon_y \\ \epsilon_{xy} \end{bmatrix} \tag{2.14}$$

where μ is the poisson's ration of the material
E is the modulus of elasticity

2.6 Strain Displacement Relationship

The displacements at the node are solved in matrix format. The displacement matrix for the structure is given below. The stresses and strains in the element can be obtained using the equation 2.15.

$$\{\epsilon\} = [B]'d \tag{2.15}$$

where B' is obtained by the following equation 2.16.

$$\{b\}' = \begin{bmatrix} a_1 & 0 & a_2 & 0 & a_3 & 0 \\ 0 & b_1 & 0 & b_2 & 0 & b_3 \\ b_1 & a_1 & b_2 & a_2 & b_3 & a_3 \end{bmatrix} \tag{2.16}$$

where a_i, b_j are obtained using the following below equation 2.17 and 2.18

$$a_1 = \frac{1}{2A}(y_2 - y_3); a_2 = \frac{1}{2A}(y_1 - y_2); a_3 = \frac{1}{2A}(y_1 - y_3); \tag{2.17}$$

$$b_1 = \frac{1}{2A}(x_2 - x_3); b_2 = \frac{1}{2A}(x_1 - x_2); b_3 = \frac{1}{2A}(x_1 - x_3); \tag{2.18}$$

$$\{d\} = \begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ . \\ . \\ . \\ u_9 \\ v_9 \\ u_{10} \\ v_{10} \end{bmatrix} \tag{2.19}$$

with boundary conditions, due to the support conditions.

$$u_9 = v_9 = u_{10} = v_{10} = 0 \tag{2.20}$$

Chapter 3

Structure, Loading conditions and Coordinate system

3.1 Structure

The structure chosen for finite element analysis is strip footing. Strip footing has its length much greater that its width and its thickness. Hence, plane strain condition can be incorporated. Below is the 3D figure 3.1 of strip footing. Strip footing is usually used as a foundation where multiple columns are present at a distant which will cause the pressure bulb of the columns overlap. The structure in figure 3.1 is assumed as the below figure 3.2 for analysis in plane strain condition.

3.1.1 Loading conditions

The basic assumptions while solving the 2D section of strip footing is:-

- Load acting from the column is acting on the node of the assumed 2D section.
- The unit weight of the soil $\gamma_d = 16kN/m^3$
- The column load acting on the footing is less than the bearing capacity of the soil and is also less than the failure load of concrete
- No reinforcements are provided to the footing
- The support conditions are assumed to be pin. Hence, displacements, u and v are resisted.

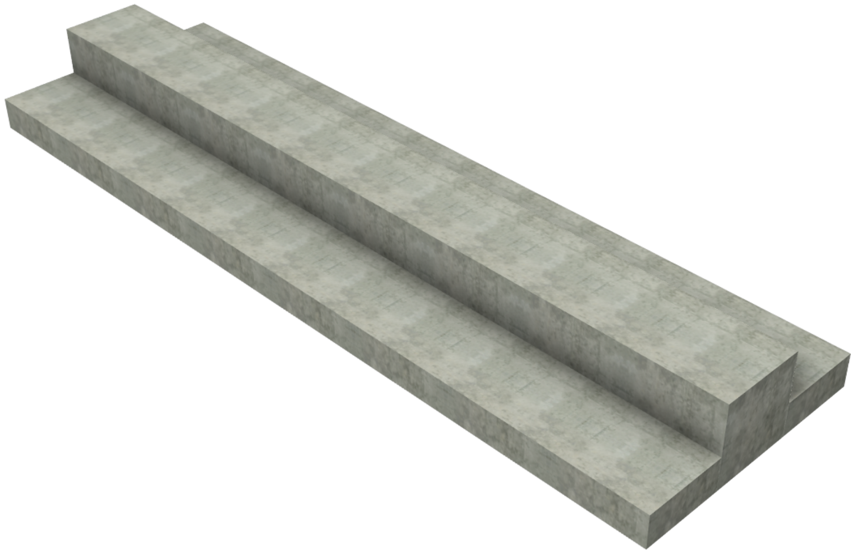

Figure 3.1: Isoparametric view of the structure

3.2 Material

For strip footing, concrete with M40 grade has been used. The poisson ratio of high strength concrete is approximately equal to 0.1.
Therefore, $\mu_{concrete} = 0.1$.
To evaluate the young's modulus of concrete, IS.456-2000 specified the below given equation. The main assumption is concrete member acts as an elastic member.

$$E_{concrete} = 5000\sqrt{f_{ck}}(MPa) \tag{3.1}$$

3.3 Plane Strain Condition

As discussed in chapter 2, the plane strain condition is applied to obtain the stiffness matrix for the element. Below is the obtained C matrix for the problem with μ =0.1 and E=31622776.6 kN/m^2

$$[C] = \begin{bmatrix} 32341476.07 & 3593497.3 & 0 \\ 3593497.3 & 32341476.07 & 0 \\ 0 & 0 & 28747978.73 \end{bmatrix} \tag{3.2}$$

Figure 3.2: Orthogonal projection of the structure for analysis

3.4 Loading conditions

The loads that were acting on the structure are given below:

- Nodal loads
  - Load acting on the column
- Line loads
  - Self weight of the structure
  - Soil above the structure acting as a surcharge

3.5 Global and local coordinate system

$$\{d\} = \begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ . \\ . \\ . \\ u_9 \\ v_9 \\ u_{10} \\ v_{10} \end{bmatrix} \tag{3.3}$$

The below shows the local coordinates system used to solve the problem.

$$\{d\} = \begin{bmatrix} d_1 \\ d_2 \\ d_3 \\ d_4 \\ d_5 \\ . \\ . \\ . \\ d_{17} \\ d_{18} \\ d_{19} \\ d_{20} \end{bmatrix} \tag{3.4}$$

Chapter 4

Results

4.1 Mesh

The below figure shows the meshing of the structure. The footing has been divided into 10 elements. The finite element analysis of the structure in 2D plane has been performed.

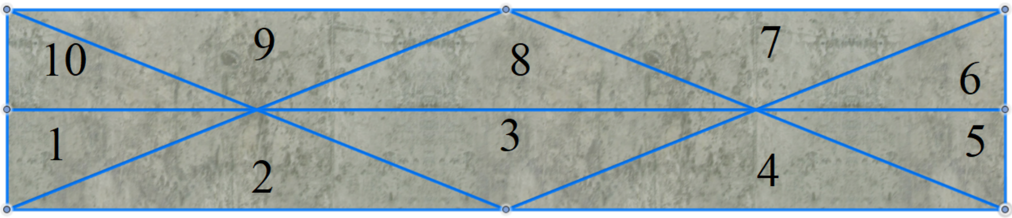

Figure 4.1: The element view of the 2D structure

4.2 Reactions

The structure is assumed to be pin supported. The reactions that are observed at the support are given below in the table 4.1

Direction	**Node**	**Reactions in (kN)**
x	1	0
y	1	0
x	2	0
y	2	0
x	3	0
y	3	0
x	4	0
y	4	0
x	5	0
y	5	0
x	6	0
y	6	0
x	7	0
y	7	0
x	8	0
y	8	0
x	9	59.66
y	9	-291.2
x	10	228.86
y	10	-150.08

Table 4.1: Support reactions of the structure.

4.3 Displacement Matrix

The obtained displacement matrix obtained (in m) is shown below.

$$\{d\} = \begin{bmatrix} 2.56112E-07 \\ 1.82203E-06 \\ -4.0576E-06 \\ -2.98378E-06 \\ -2.59424E-07 \\ -8.67899E-07 \\ -2.50008E-07 \\ 1.45339E-05 \\ -4.10113E-06 \\ -7.60654E-07 \\ -2.62102E-06 \\ 1.65571E-07 \\ -2.01987E-06 \\ 5.04934E-06 \\ 2.60896E-06 \\ 3.83445E-06 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix} \tag{4.1}$$

4.4 Stresses and strains

Stresses and strains for each element has been obtained using the 2.15 and 2.14 below are the stresses and strains of each element. The pictorial representation of the elements is given the figure. The table 4.2 provides the strains information corresponding to the element

Element Number	ϵ_{xx}	ϵ_{yy}	γ_{xy}
1	5.80358E-08	1.98918E-06	-2.59095E-07
2	-1.70742E-07	-1.0024E-07	1.43403E-06
3	-9.86738E-07	1.52343E-06	4.66268E-08
4	-1.70742E-07	4.96962E-07	6.18032E-07
5	-3.1488E-06	-5.786E-07	1.20501E-06
6	-3.84059E-06	1.9777E-06	6.29921E-07
7	1.57264E-06	3.67685E-06	9.54521E-06
8	-9.86738E-07	-9.3805E-06	4.25575E-08
9	-1.51325E-06	-3.66861E-06	9.73143E-06
10	5.80358E-08	-1.37705E-06	1.08749E-06

Table 4.2: Table of strains corresponding to the element

The below table 4.3 shows the stresses corresponding to the elements.

Element Number	$\sigma_{xx}(inkPa)$	σ_{yy} (in kPa)	τ_{xy} (in kPa)
1	9.025088798	64.54169301	-3.724229559
2	-5.882248545	-3.855480384	20.61270883
3	-26.43811471	45.72406542	0.670212602
4	-3.736203253	15.45892725	8.883591766
5	-103.915941	-30.02795958	17.32082956
6	-117.1035968	50.16065459	9.054482057
7	64.07410744	124.5660485	137.2026969
8	-65.62135143	-306.9250651	0.611720397
9	-62.12398913	-124.0861087	139.8794928
10	-3.07144633	-44.32712314	15.6315145

Table 4.3: Table of stresses corresponding to the element

4.5 Deformation

With the displacements obtained, the deformed shape of the structure can be obtained.

4.6 Stress and Strain contours

Below figure 4.2 shows the stress contours of the structure. Below figure 4.3 shows the strain contours of the structure.

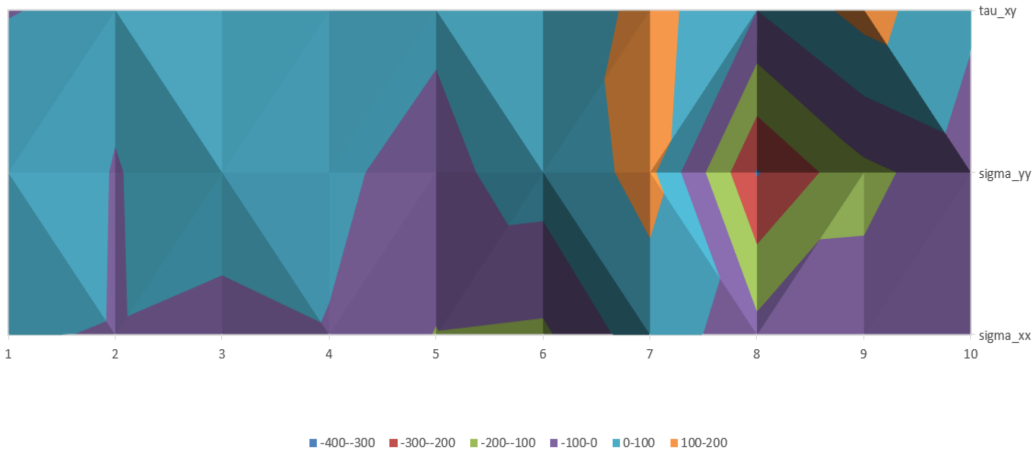

Figure 4.2: Stress contour of the structure

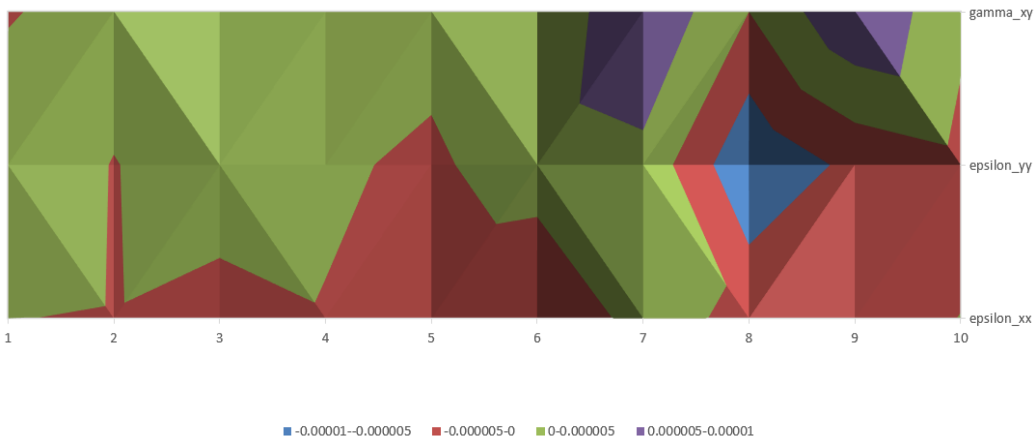

Figure 4.3: Strain contour of the structure

Chapter 5

Conclusion and Observations

It can be observed that σ_{xx} is highest in element in 6 and is in compression. The sign convention observed is Tensile as positive and compression as negative. σ_{yy} is highest in element 8 and is in tension. High distortions are observed in element 7 and element 9 which can be explained by the higher value of τ_{xy}. As the dimensions of the structure is small, and number of elements are 10, lower displacements or deformations are observed.
The strains observed are in order of $10^{-5} to 10^{7}$ which is relatively less. This is due to the loading conditions and the dimensions of the structure. As the structure is relatively small, and the loads are not loaded to the ultimate stress level. The structure is loaded in proportionality limit. Below table gives the information about the coordinates of the deformed body.

Coordinates	Deformation	**Before loading**	**After loading**
x1	2.56112E-07	1.5	1.499999744
y1	1.82203E-06	0	-1.82203E-06
x2	-4.0576E-06	0	4.0576E-06
y2	-2.98378E-06	1.5	1.500002984
x3	-2.59424E-07	3	3.000000259
y3	-8.67899E-07	1.5	1.500000868
x4	-2.50008E-07	1.5	1.50000025
y4	1.45339E-05	3	2.999985466
x5	-4.10113E-06	0.75	0.750004101
y5	-7.60654E-07	1.5	1.500000761
x6	-2.62102E-06	2.25	2.250002621
y6	1.65571E-07	1.5	1.499999834
x7	-2.01987E-06	0	2.01987E-06
y7	5.04934E-06	3	2.999994951
x8	2.60896E-06	3	2.999997391
y8	3.83445E-06	3	2.999996166
x9	0	3	3
y9	0	0	0
x10	0	0	0
y10	0	0	0

Table 5.1: Table showing the x,y coordinates before and after loading

Appendices

Appendix A

Symbols and definitions

σ - Stresses
σ_{xx} - Normal stress in x-y plane in x-direction as shown in figure A.1
σ_{yy} - Normal stress in x-y plane in y-direction as shown in figure A.1
τ_{xy}- shear stress or tangential stress in x-y plane

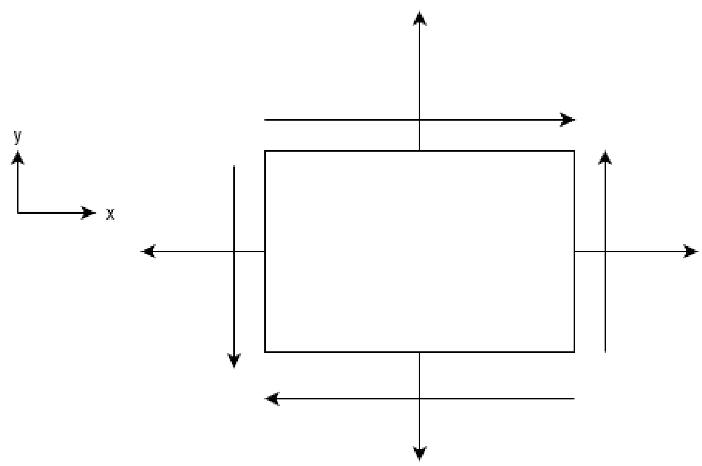

Figure A.1: Stresses in 2D plane

$E_{concrete}$=Modulus of elasticity of concrete
$\mu_{concrete}$- Poisson's ratio of concrete
f_{ck}- Characteristic strength of concrete
N - shape function
ς, s, t - Isoparametric coordinates
u- Displacements in x-direction
v-Displacements in y direction
ϵ - Strains

G- Shear modulus of the material
h- thickness of the section
A- area of the element
V_N - Nodal force vectors
$F_{i,j}$ - Force on the node i in direction j x_i - x coordinate of i^{th} node
y_i - y coordinate of i^{th} node
γ_d - Unit weight of soil
γ_{xy} - Shear strain in x-y plane
ϵ_{xx} - Normal strain in x-y plane in x-direction
ϵ_{yy} - Normal strain in x-y plane in y-direction
$\{d\}$ - Displacement vector of the structure
$\bar{\bar{C}}$= Stiffness tensor of that material
$\bar{\bar{D}}$= complaince tensor of that material