Abstract

Introduction

**FEM**

The finite element method (FEM) is a numerical procedure for solving field problems. In this course, the problems to be analyzed are considered as continuum mechanics problems.

Continuum mechanics is the study of mechanical behavior of body (solid or fluid) under the action of forces on the macroscopic scale. It ignores the discrete nature of matter, and treats material as uniformly distributed throughout region of spaces to be able to define field quantities such as density, displacement, velocity, etc. as continuous (at least piecewise) function of positions and time.

* Computational domain is discretized into a finite number of elements (which can be of different sizes and geometry).
* Each element satisfies the governing equations of the problem as well as the continuity requirements.
* Suitable for problems in structural mechanics (beams, plates, shells) and also nonlinear problems.
* May require a large number of elements for problems with infinite media (e.g. geotechnical engineering problems).

**Steps in Finite Element Analysis**

* Discretize computational domain into a finite number of elements.
* Select element shape functions for each element.
* Formulate finite element equations for each element.
* Assemble all element equations to obtain the global equation.
* Apply boundary conditions of the problem to the global equation.
* Solve the global equation for unknown parameters.
* Compute other interested quantities.

An element consists of 4 components

1. Element configuration (or shape).
2. Number and type of nodes.
3. Number of degree of freedom (DOF)/node.
4. Shape function.

**Element Configuration**

Diagram, shape

Description automatically generated

**Nodes**

Chart

Description automatically generatedNode is a point (or joint) where coordinates of position are prescribed, and field variables are solved.

There are two types of nodes

1. Exterior nodes.
2. Interior nodes.

**Degree of Freedom (DOF)**

Degree of freedom (DOF) is the primary unknown of the problem. It must be independent. If they are related to some physical meaning, then are referred as nodal variables.

Types and number of DOF depend on

1. The type of problem.
2. The space of problem.
3. The formulation method.

**Shape Function**

Shape function, also known as interpolation function or trial function, is employed in FEM to represent.

1. Geometry of the element domain.
2. Variation of fields over domain.

* If order of the shape function of (1) > (2), superparametric element.
* If order of the shape function of (1) < (2), subparametric element.
* If order of the shape function of (1) = (2), isoparametric element.

Model 2D

**Bassic Equation**

a) Euler- Bernoulli Beam element:

Input

 

Supply the element nodal coordinates x1, y1, x2, and y2, the modulus of elasticity E, the cross section area A, the moment of inertia I.

Stiffness matrix







The transformation matrix G contains the direction cosines

 

Where the length



Theory

The evaluation of the section forces is based on the splutions of the basic equations

 

From these equations, the displacements along the beam element qre obrained the sum of the homogeneous and the particular solutions



Where

 

And

  

The transformatrix Ge and nodal displacement ae are described in feframe2 .Note that the transpose of ae is stored ed.

Finally the section forces are otained from

  

b) Timoshenko Beam element:

Input

 

Supply the element nodal coordinates x1, y1, x2, and y2, the modulus of elasticity E, the shear modulus G, the cross section area A, the moment of inertia I and the shear corretions factor ks.

Stiffness matrix





With



And



The transformation matrix G contains the direction cosines

 

Where the length



Theory

The evaluation of the section forces is based on the solutions of the basic equations

  

( The equa tions are valid if is not more than a linear function of ). From these equations, the displacement slong the beam element are obtained as sum of the omogenous and the particular solutions.



Where

 

And

 

 

The transformation matrix G and nodal displacements ae are described in feframe2.

Fimally the section forces are obtained from

  

c) Mass matrix

Input

 

Contains the modulus of elasticity E, the cross section area A, the moment of inertia I,

The spring stiffness in the axial direction ka, and the spring stiffness in the transverse direction kt.

The element load vector fe can also be compurted if unifromly distributed loads are applies to the element. The optional input variable eq, dercibed in feframe2, contains the distributed load.

Theory

The elememt stiffness matrix , stored in ke, is computed according to



Where





The matrix is given by



Where the length



**Function**

feframe2() Tạo ma trận độ cứng phần tử

feasmbl1() Ghép vào ma trận độ cứng tổng thể

feaplyc2() Áp điều kiện biên

extract() Chuyển ma trận chuyển bị nút thành ma trận chuyển vị tổng thể khung

beam2s() Tính toán giá trị nội lực

eldraw2() Chuyển vị vẽ mô hình từng phần tử khung

eldisp2() Vẽ đường chuyển vị từng phần tử khung

beam2crd() Tính chuyển vị từng phần tử theo Euler-Bernoulli

eldia2() Tìm tỉ lệ hợp lý thể hiện biểu đồ lực cắt ( momen)

eldia22() Vẽ biểu đồ lực cắt theo tỉ lệ

Problem

**Input**

A 400x400

L 6000mm

H 3500mm

E 32500N/mm2

Xi (I) 4/1875 mm4

rh0 (g) 2500N/mm3

nel 6

nnode 6

nnel 2

ndof 3

sdof 18

kk 18x18

ff 18x1

element 6x2

 Ma trận 1xn chứa tọa độ từng nút

 Ma trận 3xn chứa các giá trị N,V,M

 Ma trận 2xn chứa chuyển vị phần tử theo từng phương

a) Static

b) Free

c) Transient

Model 3D

**Bassic Equation**

a) Euler- Bernoulli Beam element:

Input

 

Supply the element nodal coordiates x1, y1, etc. as well as the direction of the loacal beam coordiate system . By giving a blobal vector  parallel with the positive local z axis of the beam, he local beam coordiate system is defined.The variable



Spplies the modulus of elasticity E, the shear modulus G, the cross section area A, the moment of inertia with respect to theaxis Iy, the moment of inertia with respect to theaxis Iz , and St Venant torsinal stiffness Kv.

Theory

The element stiffness matrix Ke is computed according to



Where



In which and , and where



The transformation matrix G contains the direction cosines

  

Where



b) Timoshenko Beam element:

Input

 

Supply the element nodal coordinates x1, y1,z1 x2,y2 and z2, the modulus of elasticity E, the shear modulus G, the cross section area A, Izz;Iyy the second moments of area about the z-axis, Jρ the polar moment of area and the y-axis the moment of inertia I and the shear corretions factor ks ( typically 5/6 for rectangular section).

Theory

The element stiffness matrix Ke is computed according to



Where

With

 

 

And

 

The method to determine **T** is discussed as below. Firstly, we need to slove the components of axial vector **i** in the global reference system. Given the global coordinates of the beam (*x*1; y1; *z*1; *x*2; y2; *z*2), the first row of **T** is

  

Where



Then given the cross-section direction vector in global reference system **k**’ = (*k’*1; *k’*2; *k’*3) (This is define in the input file along with the second moment of area *I*yy; *I*zz), the local (xz)-plane is formed by vectors **i** and **k**’. we can slove  
vector **j** = -**i** × **k**’, thus the second row of **T** is

  

Where



Now the normal vector **k** of local (xy)-plane is given by **k** = **i** × **j**. Therefore the third row of **T** is given by

  

Where



Theory

The evaluation of the section forces is based on the solutions of the basic equations

  

( The equa tions are valid if is not more than a linear function of ). From these equations, the displacement slong the beam element are obtained as sum of the omogenous and the particular solutions.



Where

 

And

 

 

The transformation matrix G and nodal displacements ae are described in feframe2.

Fimally the section forces are obtained from

  

Problem

+ Static

+ Free

+ Transient