STATIC ANALYSIS OF CONTINUOUS BEAM WITH NUMERICAL METHOD (FEM)

ANALIZA STATICĂ A GRINZILOR CONTINUE FOLOSIND METODA NUMERICĂ (FEM)

FETEA M.[1], ZAHARIA A.[2]
e-mail: marius_fetea@yahoo.com

Abstract. *Finite element method is a method of analysis and simulation of current real phenomena. This paper focuses on this method, applied through finite element analysis program Matlab, presenting a structural analysis application useful in the field of forest, mechanical and structural engineering. Program designed by the authors using the finite element tool engineer put in hand work necessary to optimize the design, with positive effects on the complete analysis of stress and tensions in continuous beams.*
Keywords: Engineering, finite, element, beam, methods.

Rezumat. *Metoda elementului finit reprezintă metoda de analiză şi simulare a fenomenelor reale. Lucrarea bazată pe metoda enunţată şi programul de calcul nemeric Matlab, prezintă posibilitatea de efectuarea a analizelor structurale în domeniul forestier, mecanic şi civil. Programul conceput de către autori, folosind metoda elementului finit, permite optimizarea muncii de proiectare, având efecte pozitive în ceea ce priveşte analiza tensiunilor şi eforturilor ce apar în secţiunile transversale ale grinzilor continue.*
Cuvinte cheie: Inginerie, element finit, metodă, grindă.

INTRODUCTION

In the finite-element method, a distributed physical system to be analysed is divided into a number (often large) of discrete elements. The division into elements may partly correspond to natural subdivisions of the structure. Most or all of the model parameters have very direct relationships to the structure and material properties of the system (Fletcher, 1959; Gheorghiu, 1999; Glazman and Liubici, 1980)

MATERIAL AND METHOD

This paper presents the calculation of flat structures with rigid nodes using finite element method. In this case there are no inertial or damping effects, or at least they are negligible (Bors, 2007; Jianming and Douglas, 2009; Leissa, 1962).

Flat structure is modeled as a simple continuous beam with simply supports and 1 articulated support located at the left end of the structureThis type of structure is composed of bars whit 2 nodes and 3 degrees of freedom on each node (Barsan, 1983; Ille and Soare, 1983; Fletcher, 1959; Gheorghiu,1999).

[1]University of Oradea, Romania
[2] University of Agricultural Sciences and Veterinary Medicine of Cluj-Napoca, Romania

The three degrees of freedom per node are the horizontally and vertically displacements and also the rotated section. It aims to determine the nodals elastic equilibrium equations using the displacements method (Catarig et al.,1978; Manescu, 2005; Timoshenko and Goodier, 1970; Szilard, 1974). The analysis requires two reference systems (Pantel, 2002; Petrila,1987) one local that is attached to each element of the bar and a global for the analysis of the entire structure of bars.

RESULTS AND DISCUSSION

Generalized displacement and generalized forces vectors of a beam element are (Fletcher, 1959; Fetea, 2010; Bors, 2007).

$$\{d\} = \{u_i, v_i, \varphi_i, u_j, v_j, \varphi_j\}^T, \{f\} = \{N_i, T_i, M_i, N_j, T_j, M_j\}$$

Stiffness matrix elements are determined by applying displacements on each degree of freedom and blocking the corresponding the other remaining degrees of freedom. At each applied displacement nodes produce at the ends of bar sectional efforts on the 6 degrees of freedom (Jianming and Douglas, 2009; Pantel, 2002). By applying the 6 successive displacements and using the principle of superposition, determine the relationship between generalized displacement and generalized forces vector . Stiffness matrix contains terms that depend on the geometry of the beam and physical-mechanical properties of the material.

Orthogonal matrix that connects the components of a vector in global and local reference system is a transformation matrix and is of the form (Glazman and Liubici 1980; Manescu, 2005).

The displacement and efforts at the ends of bars is determinate by applying conditions and solving the system equations of the nodal equilibrium. By applying the superposition principle (Via et al., 1983; Catarig and Petrina, 1978), we determined the relation between the sectional efforts to ends beam, when were applied nodal displacements (u, v, φ) in each node on the 3 degrees of freedom. This is the equilibrium equation of beam elements in the local reference system.

Initial data of beam studies are: force is applied to the beam in node 8 having the coordinates $x = 800[mm], y = 0[mm]$; Section height is $h = 100[mm]$; Young's modulus $E = 2.1 \cdot 10^5 [N/mm^2]$; Transverse modulus of the material $E = 8 \cdot 10^4 [N/mm^2]$;Tensile-compressive stiffness of the structure $E \cdot A = 100^2$; Bending stiffness of the structure. $E \cdot I_z = 100^4 / 12$, $F_y = 10^3 [N]$.

The numerical program.

```
% Continuous beam is considered. Required to determine the nodal displacements, stresses and sectional efforts at the ends of bars.
nodes= [0   0
        200 0
        400 0
        600 0
        800 0
```

```
clear; clc;
%Cartesian coordinates of the nodes expressed in [mm]
%        x      y

        1000 0
        1200 0
        1400 0
        1600 0
        1800 0
```

```
        2000 0]
% Finite element matrix
elem=[ 1    2    100
          2    3    100
          3    4    100
          4    5    100
          5    6    100
% Young's modulus [N/mm^2]
E=2.1*10^5
% Transverse modulus of the material [N/mm^2]
G=8*10^4
% Tensile-compressive and bending stiffness of the structure.
ea=100^2 eiz=100^4/12
%Number of nodes of the structure
nnd=length(noduri(:,2))
cond=[ 1   1   1   0
          3   0   1   0
          5   0   1   0
% Determine the number of forces and boundary conditions applied to the structure
nnf=length(forte(:,1))
ncond=length(cond(:,2))
% Axes x and y coordinates of the node structure
cx=noduri(:,1) cy=noduri(:,2)
%Number of degrees of freedom per node (ngn),element (nel) and the total number of degrees of freedom (nec)
ngn=3 ngel=2*ngn nec=nnd*ngn
% Initialization to zero for MR, F and index
MR=zeros(nec,nec)            F=zeros(nec)
index=zeros(2*ngn)
% The calculation of the beam with rigid nodes
for i=1:nel
mrel=[e1  0   0  e1  0   0
        0  e2  e3  0  -e2  e3
        0  e3  e4  0  -e3  e5
       -e1  0   0  e1  0   0
        0 -e2 -e3  0   e2 -e3
        0  e3  e5  0  -e3  e4]
% Transformation matrix
```

```
     % elem nod1 nod2  h(section height)
          6    7    100
          7    8    100
          8    9    100
          9   10    100
         10   11    100]
% Number of elements of structure
nel=length(elem(:,2))
% Forces applied to the beam
%     node fx   fy   momz
forte=[  8   0  -1000    0]
% Boundary conditions applied to the beam
%     node bx by  brz


          7   0   1   0
          9   0   1   0
         11   0   1   0]
nod1=elem(i,1)              nod2=elem(i,2)
h(i)=elem(i,3)
for ii=1:ngn
index(ii)=ngn*(nod1-1)+ii
end
for iii=ngn+1:2*ngn
index(iii)=ngn*(nod2-2)+iii
end
% Length of beam finite elements
le=sqrt((cx(nod2)-cx(nod1))^2+(cy(nod2)-cy(nod1))^2)
% Cosines directors of beam elements.
c=(cx(nod2)-cx(nod1))/le s=(cy(nod2)-cy(nod1))/le length(i)=le'
% Vectors cosine directors
vc(i)=c vs(i)=s
% Matrix elements stiffness
e1=ea/le   e2=12*eiz/le^3   e3=6*eiz/le^2
e4=4*eiz/le e5=2*eiz/le
c1=[c -s 0]' c2=[s c 0]' c3=[0 0 1]'  c0=[0 0 0]'
T=[c1 c2 c3 c0 c0 c0
     c0 c0 c0 c1 c2 c3]
% Stiffness matrix in global reference system
mrel=T'*mrel*T
```

```
% Assembling the stiffness matrices of elements
for i1=1:ngel
j1=index(i1)
for i2=1:ngel
j2=index(i2)
MR(j1,j2)=MR(j1,j2)+mrel(i1,i2)
% Set up vector of nodal loads
for i=1:nnf
%Forces  nodes
n=forte(i,1)
if forte(i,2)~=0
f=forte(i,2)
n=cond(i,1) end
% Implementation of the conditions with zero displacement on  x direction
if cond(i,2)==1
MR(ngn*(n-1)+1,:)=zeros(1,nec)
MR(:,ngn*(n-1)+1)=zeros(nec,1)
MR(ngn*(n-1)+1,ngn*(n-1)+1)=1
F(ngn*(n-1)+1)=0
end
% Implementation of the conditions with zero displacement on  y direction
if cond(i,3)==1
MR(ngn*(n-1)+2,:)=zeros(1,nec)
MR(:,ngn*(n-1)+2)=zeros(nec,1)
MR(ngn*(n-1)+2,ngn*(n-1)+2)=1
F(ngn*(n-1)+2)=0
end
% Implementation of the conditions with zero rotations around  z axes
if cond(i,3)==1
MR(ngn*(n-1)+3,:)=zeros(1,nec)
MR(:,ngn*(n-1)+3)=zeros(nec,1)
MR(ngn*(n-1)+3,ngn*(n-1)+3)=1
F(ngn*(n-1)+3)=0
end
% Determination of initial unknowns represented by nodal displacements by solving the system of elastic nodal equations
% Format long e
depl=MR\F
for i=1:nnd
u(i)=depl(ngn*(i-1)+1)  v(i)=depl(ngn*(i-1)+2) rotz(i)=depl(ngn*(i-1)+3)
F(ngn*(n-1)+1)=F(ngn*(n-1)+1)+f
end
if forte(i,3)~=0
f=forte(i,3)
F(ngn*(n-1)+2)=F(ngn*(n-1)+2)+f
end
if forte(i,4)~=0
f=forte(i,4)
F(ngn*(n-1)+3)=F(ngn*(n-1)+3)+f
end
% Applying boundary conditions
for i=1:ncond
% Nodes with displacement zero
end
% Display the primary unknowns (nodal displacements)
fprintf('nod u(mm) v(mm) rotz(rad)\n')
for i=1:nnd
fprintf(' %2.f %2.5f
%2.5f\n',i,u(i),v(i),rotz(i))
end
fprintf('\n')
pause
%Determination of strains and tensions in ends of each beam finite element
for i=1:nel
% Redefining nodes
nod1=elem(i,1) nod2=elem(i,2)
% Calculation of beam lengths of all finite elements
le=sqrt((cx(nod2)-cx(nod1))^2+(cy(nod2)-cy(nod1))^2)
% Determine the cosine directors of each beam finite element
c=(cx(nod2)-cx(nod1))/le s=(cy(nod2)-cy(nod1))/le
% Determination of global displacement of each beam finite element
ue1=depl(elem(i,1)*ngn-2,1)
ue2=depl(elem(i,1)*ngn-1,1)
ue3=depl(elem(i,1)*ngn,1)
ue4=depl(elem(i,2)*ngn-2,1)
ue5=depl(elem(i,2)*ngn-1,1)
ue6=depl(elem(i,2)*ngn,1)
%Determination of nodal displacements for each beam finite element in local reference system
```

```
ul1=c*ue1+s*ue2      ul2=(-s)*ue1+c*ue2
ul3=ue3       ul4=c*ue4+s*ue5       ul5=(-
s)*ue4+c*ue5 ul6=ue6
% Calculation of stress from the first end
of the beam
% Strain from tensile (compressive)
e11=(ul4-ul1)/le
% Strain from the bending deformation
e12=h(i)/(2*le^2)*(-6*ul2-
4*ul3*le+6*ul5-2*ul6*le) stress(1)=(e11-
e12)*elem(i,3)
stress(2)=(e11+e12)*elem(i,3)
%The  maximum stress to the first end of
the beam
STRESS1=max(abs([stress(1),stress(2)]))
% Calculation of stress at the second end
of the beam
% Strain from tensile (compressive)
e21=(ul4-ul1)/le
% Strain from the bending deformation
e22=h(i)/(2*le^2)*(6*ul2+2*ul3*le-
6*ul5+4*ul6*le) stress(3)=(e21-
e22)*elem(i,3)
stress(4)=(e21+e12)*elem(i,3)
% The  maximum stress to the first end of
the beam
STRESS2=max(abs([stress(3),stress(4)]))
%Maximum stress on each finite element
beam
Stresselem=max(abs([STRESS1,STRESS
2]))  tensmax(i)=Stresselem
mrel=[e1  0   0  e1  0   0
        0  e2  e3  0  -e2  e3
        0  e3  e4  0  -e3  e5
% Transformation matrix of elements
c1=[c -s 0]' c2=[s c 0]' c3=[0 0 1]' c0=[0
0 0]'
T=[c1 c2 c3 c0 c0 c0
  c0 c0 c0 c1 c2 c3]
% Stiffness matrix in global reference
system
mrel=T'*mrel*T
% The vector displacement for finite
element beam
deplel=[u(nod1,v(nod1,rotz(nod1),u(nod2
),v(nod2),rotz(nod2)]'
% Sectional efforts vector
% Display nodal unknowns tension and
maximum stress on each beam finite
elements
fprintf('\n')
fprintf('element  %2.f\n',i)
fprintf('node %2.f stressnod1 stressnod2
maxstressnode1 \n',elem(i,1))
fprintf(' %2.5f  %2.5f
%2.5f\n',stress(1),stress(2),STRESS1)
fprintf('node %2.f stressnod2 stressnod2
maxstressnode2 \n',elem(i,2))
fprintf(' %2.5f  %2.5f
%2.5f\n',stress(3),stress(4),STRESS2)
fprintf('Maximum stress on element)
fprintf(' %2.5f \n', Stresselem)
end
for i=1:nel
% Redefining nodes
nod1=elem(i,1) nod2=elem(i,2)
% Calculation of beam lengths of all
finite elements
le=sqrt((cx(nod2)-
cx(nod1))^2+(cy(nod2)-cy(nod1))^2)
%Determine the cosine directors of each
finite element
c=(cx(nod2)-cx(nod1))/le s=(cy(nod2)-
cy(nod1))/le
% Stiffness matrix of element
e1=ea/le e2=12*eiz/le^3 e3=6*eiz/le^2
e4=4*eiz/le e5=2*eiz/le

          -e1   0   0  e1  0   0
           0 -e2 -e3  0   e2 -e3
           0  e3  e5  0  -e3  e4]
ef=mrel*deplel nx(1,i)=-ef(1)
nx(2,i)=ef(4) ty(1,i)=-ef(2) ty(2,i)=ef(5)
mz(1,i)=-ef(3) mz(2,i)=ef(6)
% Display the sectional efforts
fprintf('\n')
fprintf('elementul %2.f\n',i)
fprintf('nod %2.f Fx Fy Mz \n',elem(i,1))
fprintf(' %2.5f  %2.5f
%2.5f\n',ef(1),ef(2),ef(3))
fprintf('nod %2.f Fx Fy Mz \n',elem(i,2))
fprintf(' %2.5f  %2.5f
%2.5f\n',ef(4),ef(5),ef(6))end
```

CONCLUSIONS

Numerical method has the advantage that the computer program developed by the author, leads to solutions of the problem that converge to the "exact" solution. The paper presented, is a novelty in terms of adapting to a full calculation of continuous beams regardless of physical-mechanical properties of materials they are made.

The main steps that were followed in this program by the authors are:

1. Stiffness matrices-writing of the elements composing the structure of the continous beam;
2. Calculation of the cosine directors and transformation matrices;
3. Matrix assembly of each beam in the global stiffness matrix of the structure;
4. Establishment of nodal forces for the entire structure;
5. Application related conditions;
6. Determining the nodal equilibrium equations system;
7. Determining the efforts and the tension at each beam ends.

REFERENCES

1. **Barsan G., 1983 -** *Mecanica-Statica*, Atelierul de multiplicare al Institutului Politehnic. Cluj-Napoca.
2. **Via C., Ille V., Soare M., 1983** - *Reziztența Materialelor și Teoria Elasticității*, Ed. Didactică și Pedagogică, București,.
3. **Bors I. 2007 -** *Aplicații ale problemei de valori proprii în mecanica construcțiilor - sisteme continue*. Editura U.T. Pres.
4. **Catarig A., Petrina M., Cocheci T., 1978** - *Mecanica Constructiilor*, Atelierul de Multiplicarea a Institutului Politehnic, Cluj Napoca.
5. **Fetea M., 2010** - *Calcul analitic și numeric în Rezistența Materialelor*. Universitatea din Oradea.
6. **Fletcher H.J., 1959** - *The Frequency of Vibration of Rectangular Isotropic Plates*, J. Appl. Mech., vol. 26, no. 2 , p. 290.
7. **Gheorghiu C. I., 1999** - *A Constructive Introduction to Finite Elements Method*, Editura Quo-Vadis, Cluj-Napoca.
8. **Glazman I. M., Liubici Iu. I, 1980** - *Analiza liniara pe spatii finit-dimensionale*, Editura Ştiințifică și Enciclopedică.
9. **Jianming J, Douglas R., 2009 -** *Finite Element Analysis of Antennas and Arrays*, Ed. Wiley.
10. **Leissa A.W., 1962** - *A Method for Analyzing the Vibration of Plates*, J. Aerospace Sci., vol. 29, no. 4, p. 475.
11. **Manescu T., 2005 -** *Analiza structurala prin metoda elementului finit*, Editura Orizonturi Universitare, Timisoara.
12. **Pantel E., 2002 -** *Lectii de Rezistenta Materialelor*, Editura Napoca Star.
13. **Petrila T., 1987 -** *Metoda elementului finit şi aplicații*, Editura Academiei RSR, Bucureşti.
14. **Timoshenko, St. P., Goodier J.N., 1970 -** *Theory of Elasticity*, 3-rd Edition, Mc Graw-Hill Book Comp, New York.
15. **Szilard R.,1974 -** *Theory and Analysis of Plates*, Prentice-HallInc., Englewood Cliffs, New Jersey.