IMK-14 – Istraživanje i razvoj u teškoj mašinogradnji 24(2018)4, SR111-116
UDC 621 ISSN 0354-6829

Statička analiza ravanskih rešetkastih nosača korišćenjem metoda krutih segmenata

Aleksandar Nikolić[1*], Goran Bošković[1*]
[1]Fakultet za mašinstvo i građevinarstvo u Kraljevu, Univerzitet u Kragujevcu

U ovom radu predstavljen je novi pristup za sprovođenje statičke analize ravanskih rešetkastih nosača. U osnovi ovog pristupa je diskretizacija elastičnih štapova, koji čine osnovni element ovih nosača, na krute segmente međusobno povezane odgovarajućim elastičnim zglobom. Imajući u vidu da su štapovi od kojih su sastavljeni rešetkasti nosači opterećeni samo u podužnom pravcu, uvedeni su translatorni zglobovi sa oprugama odgovarajuće krutosti u njima. Koristeći se osnovnim principima analitičke mehanike u radu je formiran opšti algoritam za statičku analizu rešetkastih nosača različitih oblika.

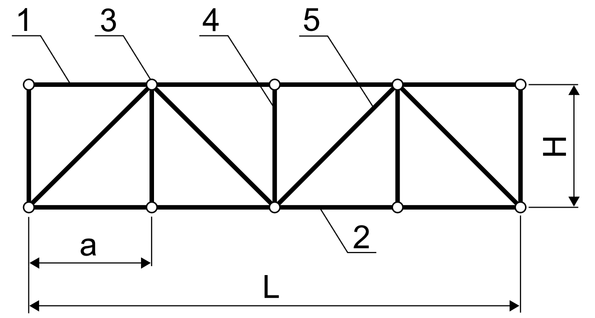
Ključne reči: Rešetkasti nosači, Statička analiza, Metod krutih segmenata

1. UVOD

Rešetkasti nosači predstavljaju sistem štapova konstantnog poprečnog preseka koji su međusobno povezani u čvorovima tako da formiraju geometrijski nepromenljive oblike.

Primenjuju se kod nosećih konstrukcija različitih namena [1], kao na primer kod hala, krovnih konstrukcija, mostova, dizalica, stubova dalekovoda, itd. Masovnost njihove primene je opravdana malim odnosom težine u odnosu na nosivost u poređenju sa punim nosačima, manjim utroškom materijala, lako se postiže uniformnost ponavljanjem istih osnovnih elemenata, i sl. Sa druge strane, proces izrade rešetkastih nosača je znatno složeniji u odnosu na pune nosače, što ponekad može dovesti u pitanje opravdanost njihove primene.

Rešetkasti nosači se mogu podeliti u dve osnovne grupe na ravanske i prostorne nosače. U ovom radu analiza će biti ograničena na ravanske rešetkaste nosače. Primer jednog ravanskog rešetkastog nosača prikazan je na slici 1. Sastoji se iz gornjeg pojasa (1), donjeg pojasa (2), čvorova (3), vertikale (4) i dijagonale (5).



Slika 1: Ravanski rešetkasti nosač

Ravanski rešetkasti nosači su kao celina najčešće opterećeni na savijanje, dok su štapovi koji čine rešetku aksijalno opterećeni, ako se pretpostavi da je veza štapova u čvorovima rešetke zglobna, a ne kruta [1]. U realnim uslovima štapovi su u čvorovima najčešće spojeni krutom vezom, što uzrokuje da su opterećeni i na savijanje. Međutim, može se pokazati da su deformacije štapova nastale usled savijanja zanemarljivo male u odnosu na aksijalne deformacije. Stoga se pri statičkoj analizi zanemaruje uticaj savijanja, tako da se pretpostavlja da su štapovi koji čine rešetku aksijalno opterećeni.

Statička analiza ove vrste nosača je neizostavan korak pri projektovanju. U tu svrhu se u novije vreme najčešće koristi metod konačnih elemenata implementiran u različitim komercijalnim softverima dostupnim na tržištu.

U ovom radu će biti prikazan jedan alternativni pristup za statičku analizu rešetkastih nosača zasnovan na metodu krutih segmenata [2]. Osnovu ovog metoda čini diskretizacija svakog elastičnog štapa rešetke, tako da se elastičnost štapa opisuje uvedenim oprugama odgovarajuće krutosti. Cilj je da se dobije univerzalni algoritam za statičku analizu ravanskog rešetkastog nosača.

2. DISKRETIZOVANI MODEL RAVANSKOG ELASTIČNOG ŠTAPA

Na slici 1(a) je prikazan diskretizovani model elastičnog štapa koji se sastoji od dva kruta štapa jednakih dužina $l_{jk}/2$, konstantnog modula elastičnosti materijala E, konstantne površine poprečnog preseka A i konstantnog momenta inercije poprečnog preseka štapa I_z. Štapovi su međusobno povezani elastičnim zglobnim elementom koji dopušta relativnu translaciju u aksijalnom i poprečnom pravcu (p_{jk} i q_{jk}), kao i relativnu rotaciju oko ose upravne na ravan kretanja $\left(r_{jk}\right)$. Krutost opruga postavljenih u zglobnom elementu JE_{jk} može se odrediti, na osnovu reference [2], na sledeći način:

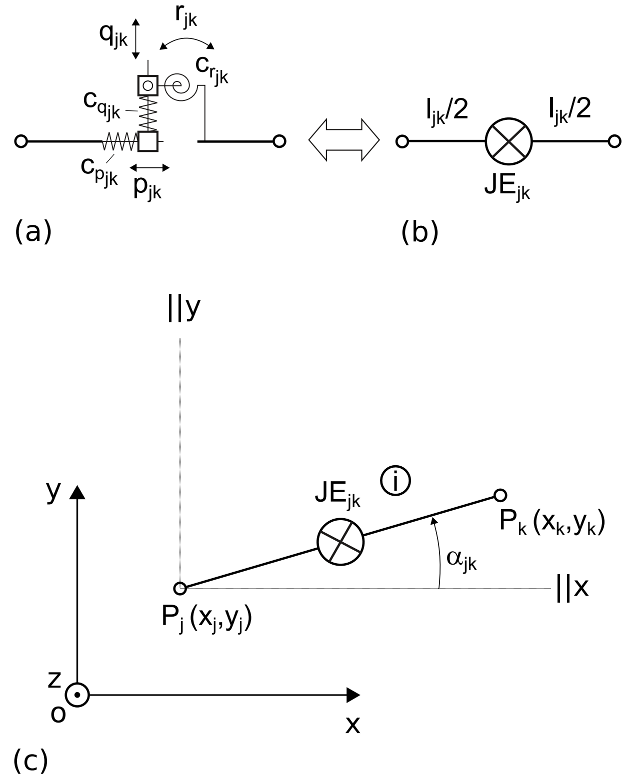
$$c_{p_{jk}}=\frac{AE}{l_{jk}},\ c_{q_{jk}}=12\frac{EI_z}{l_{jk}^3},\ c_{r_{jk}}=\frac{EI_z}{l_{jk}}, \tag{1}$$

gde je:

*Kontakt adresa autora: Fakultet za mašinstvo i građevinarstvo u Kraljevu, Univerzitet u Kragujevcu, nikolic.a@mfkv.kg.ac.rs

$$l_{jk} = \sqrt{\left(x_k - x_j\right)^2 + \left(y_k - y_j\right)^2} \tag{2}$$

dužina i-tog elastičnog štapa određena na osnovu koordinata x_j, y_j, x_k i y_k tačaka P_j i P_k u početnom ravnotežnom položaju u odnosu na izabrani nepokretni koordinatni sistem $Oxyz$. Pošto se ovde pretpostavlja da je elastični štap opterećen samo u aksijalnom pravcu, onda se može koristiti diskretizovani model elastičnog štapa koji dopušta samo pomeranje p_{jk}, dok se pomeranja q_{jk} i r_{jk} mogu zanemariti. Nadalje će predloženi model diskretizovanog elastičnog štapa biti simolički prikazan kao na slici 1(b).



Slika 2: Diskretizacija elastičnog štapa na krute segmente

3. POTENCIJALNA ENERGIJA RAVANSKOG REŠETKASTOG NOSAČA

Potencijalna energija ravanskog rešetkastog nosača može se odrediti kao suma potencijalnih energija svakog štapa koji čini rešetku ponaosob na sledeći način:

$$\Pi = \frac{1}{2}\sum_{j=1}^{n}\sum_{k=1}^{n}\gamma_{jk} c_{p_{jk}} p_{jk}{}^2, \tag{3}$$

gde je n ukupan broj čvorova u rešetki, $\gamma_{jk}\left(j<k\right)$ predstavlja identifikator koji ima vrednost 1 ako su čvorovi j i k povezani štapom ili vrednost 0 ako to nije slučaj, p_{jk} predstavlja relativno aksijalno pomeranje štapa koji povezuje čvorove j i k i može se definisati na sledeći način:

$$p_{jk} = \left(\tilde{x}_{p_k} - \tilde{x}_{p_j}\right)\cos\alpha_{jk} + \left(\tilde{y}_{p_k} - \tilde{y}_{p_j}\right)\sin\alpha_{jk}, \tag{4}$$

gde su $\tilde{x}_{p_j}, \tilde{y}_{p_j}, \tilde{x}_{p_k}$ i $\tilde{y}_{p_k}$ horizontalna i vertikalna pomeranja tačaka P_j i P_k, respektivno, u odnosu na početni ravnotežni položaj, α_{jk} predstavlja ugao koji štap i gradi sa x osom u početnom ravnotežnom položaju i važi da je:

$$\sin\alpha_{jk} = \frac{y_k - y_j}{l_{jk}},\ \cos\alpha_{jk} = \frac{x_k - x_j}{l_{jk}}. \tag{5}$$

Ako se sada uzmu u obzir izrazi (3)-(5), potencijalna energija ravanskog resetkastog nosača se može zapisati u obliku pozitivno definitne kvadratne forme na sledeći način:

$$\Pi = \frac{1}{2}\mathbf{v}^T\mathbf{K}\mathbf{v}, \tag{6}$$

gde je

$$\mathbf{v} = \left[v_1 \ldots v_j \ldots v_n\right]^T,$$

vektor pomeranja čvorova rešetke u odnosu na prvobitni ravnotežni položaj pri važi da je $v_{2i-1} = \tilde{x}_i$, $v_{2i} = \tilde{y}_i$, m predstavlja ukupan broj čvorova rešetke,

$$K = \begin{bmatrix} k_{1,1} & k_{1,2} & \ldots & k_{1,} & \ldots & k_{1,2n} \\ k_{2,1} & k_{2,2} & & k_{2,j} & & k_{2,2n} \\ \vdots & & \ddots & & & \vdots \\ k_{i,1} & k_{i,2} & \ldots & k_{i,j} & \ldots & k_{i,2n} \\ \vdots & & & & \ddots & \vdots \\ k_{2n,1} & k_{2n,2} & \ldots & k_{2n,j} & \ldots & k_{2n,2n} \end{bmatrix} \tag{7}$$

je matrica krutosti razmatrane ravanske rešetke pri čemu se koeficijenti krutosti mogu odrediti na osnovu izraza za potencijalnu energiju (3) na sledeći način:

$$k_{i,j} = \frac{\partial^2\Pi}{\partial v_i \partial v_j},\ \left(i, j = 1,\ldots,n\right). \tag{8}$$

4. USLOVI RAVNOTEŽE RAVANSKOG REŠETKASTOG NOSAČA

Pretpostavimo da na rešetkasti nosač u čvorovima u opštem slučaju deluju vektori spoljašnjih sila

$$\mathbf{F}_j = \left[F_{x,j}\ F_{y,j}\right]^T, \left(j = 1,\ldots,n\right), \tag{9}$$

gde su $F_{x,j}$ i $F_{y,j}$ projekcije vektora sile $\mathbf{F}_j$ na koordinatne ose x i y, respektivno. Ovde treba naglasiti da se u spoljasnje sile računaju i spoljasnje reakcije veza koje potiču od načina oslanjanja razmatranog rešetkastog nosača.

Sada se ukupni vektor spoljašnjih sila koje deluju na rešetkasti nosač može zapisati kao:

$$\mathbf{F} = \left[\mathbf{F}_1{}^T \ldots \mathbf{F}_j{}^T \ldots \mathbf{F}_n{}^T\right]^T. \tag{10}$$

Usled dejstva prethodno definisanog vektora spoljašnjih sila na ravanski rešetkasti nosač, dolazi do njegovog pomeranja iz prvobitnog u novi ravnotežni položaj. Koristeći Lagranžev princip virtualnih pomeranja

[3], na osnovu izraza (6) i (10), uslov koji mora biti ispunjen da bi novi položaj bio ravnotežni glasi:

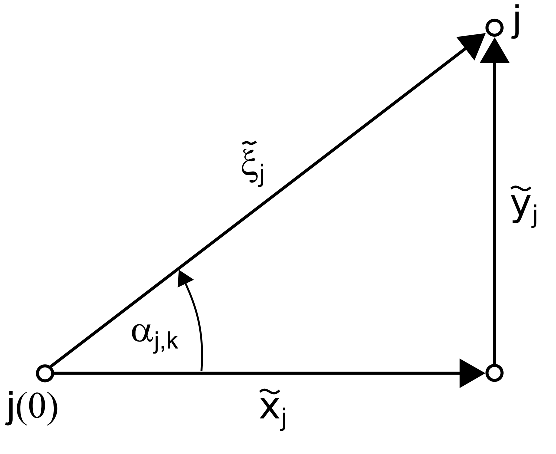
$$-\mathbf{Kv}+\mathbf{F}=\mathbf{0}_{2n\times1}. \tag{11}$$

Rešavanjem prethodnog sistema jednačina po nepoznatom vektoru pomeranja čvorova rešetke, dobija se da je:

$$\mathbf{v}=\mathbf{K}^{-1}\mathbf{F}. \tag{12}$$

Pre određivanja nepoznatog vektora pomeranja čvorova rešetke $\mathbf{v}$, potrebno je definisati konturne uslove. Zavisno od načina oslanjanja razmatranog rešetkastog nosača, moguće je da neki od čvorova rešetke budu fiksirani pokretnim ili nepokretnim zglobom za nepokretnu podlogu. U tom slučaju pomeranja u j - tom zglobu su ograničena, pa je tako $\tilde{x}_j=\tilde{y}_j=0$ u slučaju nepokrenog zglobnog oslonca, i $\tilde{x}_j=0$ ili $\tilde{y}_j=0$, zavisno od orijentacije pokretnog zgloba, tj. da li je horizontalno ili vertikalno orijentisan. Zato je potrebno u matrici krutosti $\mathbf{K}$ ukloniti redove i kolone, a u vektorima $\mathbf{v}$ i $\mathbf{F}$ samo redove, koji odgovaraju ograničenim pomeranjima.

Nakon određivanja vektora nepoznatih pomeranja čvorova rešetke $\mathbf{v}$ iz izraza (12), vraćanjem vrednosti pomeranja za svaki čvor nosača (i onih čija je vrednost jednaka nuli) u izraz (11), koji se odnosi na nosač oslobođen od veza, mogu se dobiti i nepoznate spoljašnje reakcije veza.



Slika 3: Pomeranja u j - tom čvoru rešetke

Nakon određivanja pomeranja u svim čvorovima rešetke, moguće je odrediti i sile u svim štapovima rešetke. Najpre se mogu definisati deformacije u čvorovima j i k u pravcu štapa koji povezuje te čvorove (videti sliku 3):

$$\tilde{\xi}_j=\tilde{x}_j\cos\alpha_{jk}+\tilde{y}_j\sin\alpha_{jk}, \tag{13}$$

$$\tilde{\xi}_k=\tilde{x}_k\cos\alpha_{jk}+\tilde{y}_k\sin\alpha_{jk}. \tag{14}$$

Na osnovu prethodna dva izraza, aksijalna sila koja se javlja u štapu koji povezuje čvorove j i k se može definisati kao [4]:

$$N_{jk}=\frac{EA}{l_{jk}}\left[\left(\tilde{x}_k-\tilde{x}_j\right)\cos\alpha_{jk}+\left(\tilde{y}_k-\tilde{y}_j\right)\sin\alpha_{jk}\right]. \tag{15}$$

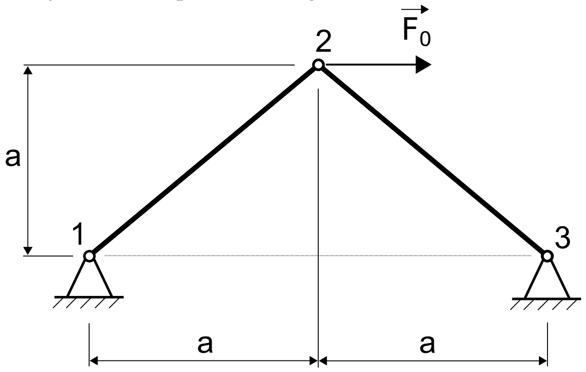
Sada se može definisati i normalni napon (pritiska ili istežanja) u svakom štapu rešetke na sledeći način [4]:

$$\sigma_{jk}=\frac{N_{jk}}{A}. \tag{16}$$

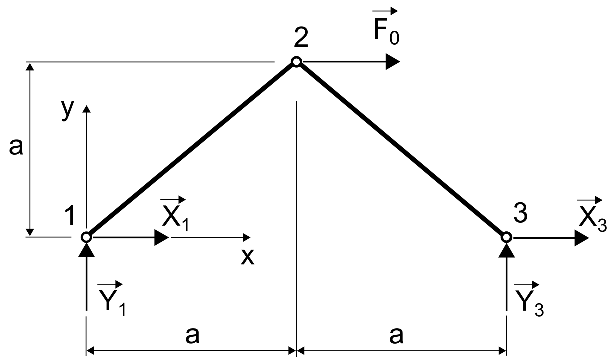
5. NUMERIČKI PRIMERI

5.1. Primer 1

Na slici 4 prikazan je sistem od dva elastična štapa dužine $\sqrt{2}a$, konstantne površine poprečnog preseka A i momenta inercije poprečnog preseka u odnosu na upravnu osu I_z koji su međusobno povezani u čvoru 2. Modul elastičnosti materijala štapova je E. U čvorovima 1 i 3 štapovi su oslonjeni na nepokretne zglobne oslonce. Ako u zglobu 2 deluje horizontalna sila konstantnog intenziteta F_0 potrebno je odrediti pomeranja u zglobu 2, kao i reakcije veza u nepokretnim zglobnim osloncima 1 i 3.



Slika 4: Rešetkasti nosač sastavljen od dva štapa



Slika 5: Rešetkasti nosač oslobođen od spoljašnjih veza

Koordinate čvorova rešetke su $x_1=y_1=0,\ x_2=y_2=a,\ x_3=2a,\ y_3=0$ a identifikatori povezanosti čvorova rešetke su $\gamma_{12}=\gamma_{23}=1,\ \gamma_{13}=\gamma_{31}=\gamma_{21}=\gamma_{32}=0$.

Korišćenjem zadatih parametara rešetkastog nosača iz izraza (1) do (8) može se odrediti matrica krutosti razmatranog rešetkastog nosača koja glasi:

$$\mathbf{K}=\frac{\sqrt{2}}{4}\frac{AE}{a}\begin{bmatrix}1&1&-1&-1&0&0\\1&1&-1&-1&0&0\\-1&-1&2&0&-1&1\\-1&-1&0&2&1&-1\\0&0&-1&1&1&-1\\0&0&1&-1&-1&1\end{bmatrix}. \tag{17}$$

Imajući u vidu izraz (10), vektor spoljašnjih sila koje deluju na razmatrani nosač je:

$$\mathbf{F}=\begin{bmatrix}X_1 & Y_1 & F_0 & 0 & X_3 & Y_3\end{bmatrix}^T. \tag{18}$$

Takođe, vektor pomeranja čvorova rešetke sada glasi:

$$\mathbf{v} = \begin{bmatrix} \tilde{x}_1 & \tilde{y}_1 & \tilde{x}_2 & \tilde{y}_2 & \tilde{x}_3 & \tilde{y}_3 \end{bmatrix}^T. \quad (19)$$

Ako se sada primene konturni uslovi da nema pomeranja u čvorovima 1 i 3, tj. važi da je $\tilde{x}_1 = \tilde{y}_1 = \tilde{x}_3 = \tilde{y}_3 = 0$, vektor pomeranja čvorova rešetke na osnovu izraza (12) sada glasi:

$$\mathbf{v} = \begin{Bmatrix} \tilde{x}_2 \\ \tilde{y}_2 \end{Bmatrix} = \frac{4a}{\sqrt{2}AE}\begin{bmatrix} 2 & 0 \\ 0 & 2 \end{bmatrix}^{-1}\begin{Bmatrix} F_0 \\ 0 \end{Bmatrix} = \begin{Bmatrix} \sqrt{2}\dfrac{aF_0}{AE} \\ 0 \end{Bmatrix}. \quad (20)$$

Ako se dobijene vrednosti pomeranja sada vrate u izraz (11) dobija se da je:

$$\begin{Bmatrix} X_1 \\ Y_1 \\ F_0 \\ 0 \\ X_3 \\ Y_3 \end{Bmatrix} = -\frac{\sqrt{2}}{4}\frac{AE}{a}\begin{bmatrix} 1 & 1 & -1 & -1 & 0 & 0 \\ 1 & 1 & -1 & -1 & 0 & 0 \\ -1 & -1 & 2 & 0 & -1 & 1 \\ -1 & -1 & 0 & 2 & 1 & -1 \\ 0 & 0 & -1 & 1 & 1 & -1 \\ 0 & 0 & 1 & -1 & -1 & 1 \end{bmatrix}\begin{Bmatrix} 0 \\ 0 \\ \sqrt{2}\dfrac{aF_0}{AE} \\ 0 \\ 0 \\ 0 \end{Bmatrix},$$

odnosno,

$$\begin{Bmatrix} X_1 \\ Y_1 \\ F_0 \\ 0 \\ X_3 \\ Y_3 \end{Bmatrix} = \begin{Bmatrix} -0.5F_0 \\ -0.5F_0 \\ F_0 \\ 0 \\ -0.5F \\ 0.5F \end{Bmatrix}. \quad (21)$$

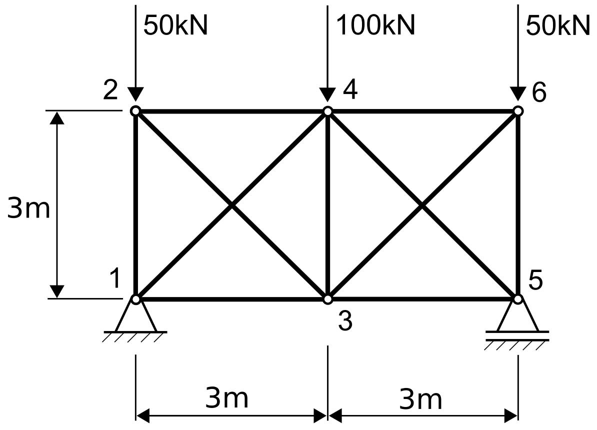
Pošto su izrazom (20) određena pomeranja u čvorovima, sada se na osnovu izraza (15) mogu odrediti aksijalne sile kao i naponi u štapovima:

$$N_{12} = -N_{23} = \frac{F_0}{\sqrt{2}},\ \sigma_{12} = -\sigma_{23} = \frac{F_0}{\sqrt{2}A}. \quad (22)$$

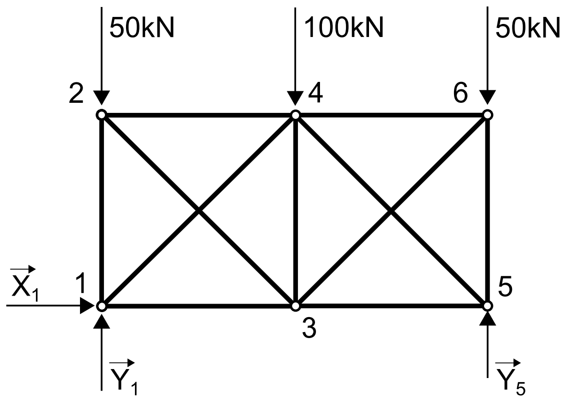
Dobijeni rezultati za pomeranja u čvorovima rešetke (20), vektor spoljašnjih sila (21), kao i za aksijalne sile i napone u štapovima (22) se u potpunosti poklapaju sa rezultatima iz reference [4], gde je za statičku analizu korišćen metod konačnih elemenata. Razlog ovog poklapanja je što se vrednost aksijalne krutosti elastičnog štapa poklapa kod oba metoda (metoda krutih segmenata i metoda konačnih elemenata).

5.2. Primer 2

U ovom primeru će biti izvršena statička analiza rešetkastog nosača prikazanog na slici 6. Nosač se sastoji od devet štapova koji su međusobno povezani posredstvom šest čvorova. U čvorovima 2, 4 i 6 deluju vertikalne sile intenziteta 50, 100 i 50 [kN], usmerene vertikalno na dole, respektivno. Nosač je sa nepokretnom podlogom spojen preko nepokretnog i pokretnog zglobnog oslonca u čvorovima 1 i 5, respektivno, pri čemu je nosač oslobođen od veza prikazan na slici 7. Dužina štapova je 3 [m], površina poprečnog preseka štapa je $A = 3 \cdot 10^{-4}\,[\mathrm{m}^2]$ a modul elastičnosti materijala štapa $E = 0.7 \cdot 10^{11}\,[\mathrm{Pa}]$.



Slika 6: Rešetkasti nosač



Slika 7: Rešetkasti nosač oslobođen od spoljašnjih veza

Koordinate čvorova rešetke su $x_1 = x_2 = 0$, $x_3 = x_4 = 3[\mathrm{m}]$, $x_5 = x_6 = 6\,[\mathrm{m}]$, $y_1 = y_3 = y_5 = 0$, $y_2 = y_4 = y_6 = 3[\mathrm{m}]$ a identifikatori povezanosti čvorova rešetke su $\gamma_{12} = \gamma_{13} = \gamma_{14} = 1$, $\gamma_{23} = 1, \gamma_{24} = 1$, $\gamma_{34} = \gamma_{35} = \gamma_{36} = 1$, $\gamma_{45} = 1, \gamma_{46} = 1$ i $\gamma_{56} = 1$. Svi ostali identifikatori povezanosti čvorova rešetke su jednaki nuli. Primenom nametnutih konturnih uslova dobija se da je $\tilde{x}_1 = \tilde{y}_1 = 0$ i $\tilde{y}_5 = 0$.

Dobijeni podaci za pomeranja u svim čvorovima rešetke prikazani su u tabeli 1. Dobijene vrednosti pomeranja su identične vrednostima dobijenim u referenci [5] gde je korišćen metod konačnih elemenata uz pretpostavku da su štapovi zglobno povezani u čvorovima. Takođe, u istoj tabeli su prikazane i vrednosti pomeranja dobijene korišćenjem softverskog paketa SAP2000 [6]. Poređenjem dobijenih rezultata može se primetiti da je razlika između dobijenih rezultata zanemarljivo mala, i ne prelazi 0.35%. Uzrok nastanka ove razlike je što je u softverskom paketu SAP2000 korišćena pretpostavka da su štapovi u zglobovima spojeni kruto, a ne zglobno kako je pretpostavljeno u ovom radu i u referenci [5]. Takođe, u tabeli 2 su prikazane sile u štapovima dobijene primenom predloženog pristupa i softverskog paketa SAP 2000. I ovde je razlika u dobijenim vrednostima zanemarljiva, maksimalno iznosi 0.64%. Bolji uvid u raspored sila u štapovim može se steći sa dijagrama prikazanog na slici 8. Mala razlika u vrednostima pomeranja u čvorovima i sila u štapovima nas navodi na zaključak da je korišćenje zglobne veze u čvorovima rešetke opravdano jer znatno pojednostavljuje analizu, a zanemarljivo malo utiče na tačnost.

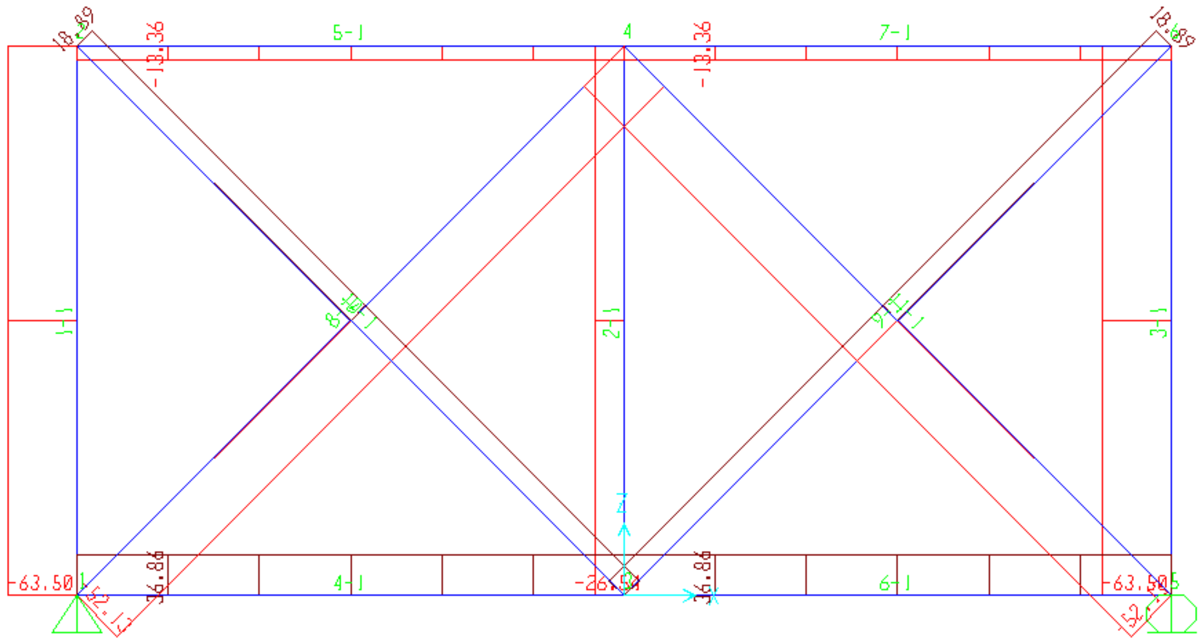
Tabela 1: Pomeranje čvorova rešetkastog nosača

Čvor	Pomeranja $\tilde{x}_i \cdot 10^3$ [m] MKS	$\tilde{x}_i \cdot 10^3$ [m] MKE	$\tilde{y}_i \cdot 10^3$ [m] MKS	$\tilde{y}_i \cdot 10^3$ [m] MKE
1	0	0	0	0
	-		-	
2	7.1429	7.1353	-9.0386	-9.0187
	0.11%		0.22%	
3	5.2471	5.2375	-16.2965	-16.2761
	0.18%		0.13%	
4	5.2471	5.2375	-20.0881	-20.0426
	0.18%		0.23%	
5	10.4942	10.475	0	0
	0.18%	0	-	
6	3.3513	3.3396	-9.0386	-9.0187
	0.35%		0.22%	

MKS - Metod krutih segmenata
MKE – Metod konačnih elemenata (SAP2000)

Tabela 2: Aksijalne sile u štapovima

N_{jk} [kN]	MKS	MKE
N_{12}	-63.27	-63.5
	0.36%	
$N_{13} = N_{35}$	36.73	36.86
	0.35%	
$N_{14} = N_{45}$	-51.94	-52.13
	0.36%	
$N_{23} = N_{36}$	18.77	18.89
	0.64%	
$N_{24} = N_{46}$	-13.27	-13.36
	0.67%	
N_{34}	-26.54	-26.54
	-	



Slika 8: Dijagram aksijalnih sila u štapovima rešetkastog nosača (SAP2000)

6. ZAKLJUČAK

U radu je prikazan jedan opšti univerzalni pristup za statičku analizu ravanskih rešetkastih nosača. Izvedeni su uslovi ravnoteže rešetkastog nosača u ravni uz korišćenje osnovnih principa analitičke mehanike. Kroz dva prikazana numerička primera je pokazana efikasnost i preciznost predloženog pristupa. Dobijeni su rezultati koji su identični rezultatima koji su dobijeni primenom metoda konačnih elemenata. Izloženi algoritam se može lako implementirati uz korišćenje čak i besplatnih programskih jezika, npr. Python, R, Octave, Scilab, Maxima, itd.

Predloženi način diskretizacije elastičnog štapa u ravni može se iskoristiti i za analizu ravanskih ramova. U tom slučaju bi se koristio zglobni element sa tri stepena slobode kretanja, koji je predložen na početku rada. Analiza ravanskih ramova će biti predmet daljeg istraživanja autora.

ZAHVALNICA

Ovaj rad je rezultat istraživanja u sklopu projekata ON174016 i TR35038 koji je finansiran od strane Ministarstva prosvete, nauke i tehnološkog razvoja Republike Srbije. Ovom prilikom izražavamo zahvalnost za podršku.

LITERATURA

[1] Z. Petković, D. Ostrić, "Metalne konstrukcije u mašinogradnji 1", Mašinski fakultet u Beogradu, Beograd, (1996)

[2] A. Nikolić, S. Šalinić, "Dynamics of the Rotating Cantilever Beam", Proceedings of IX International Conference "Heavy Machinery HM 2017", Zlatibor (Serbia), June 28-July 1, pp.D(7-12), (2017)

[3] A. I. Lurie, “Analytical Mechanics”, Springer Science & Business Media, (2013)

[4] A. Öchsner, R. Makvandi, “Finite Elements for Truss and Frame Structures: An Introduction Based on the Computer Algebra System Maxima”, Springer, (2018).

[5] A. J. FERREIRA, “*MATLAB codes for finite element analysis: solids and structures”,* Springer Science & Business Media, (2008)

[6] https://www.csiamerica.com/products/sap2000

IMK-14 – Research & Development in Heavy Machinery 24(2018)4, EN 113-118
UDC 621 ISSN 0354-6829

Static Analysis of Planar Truss by Using Rigid Segment Method

Aleksandar Nikolić[1*], Goran Bošković[1*]
[1]Faculty of mechanical and civil engineering in Kraljevo, University of Kragujevac

The new approach to the static analysis of planar truss is presented in this paper. The basis of this approach is to make a discretization of elastic bars of which truss consist on a rigid segments mutually connected by elastic joint element. Taking into account that the bars in truss are only axially loaded, the prismatic joints with springs of corresponding stiffness in them are used. The general algorithm for static analysis of trusses with various shapes was formed by using the fundamental principles of analytical mechanics.

Keywords: Truss, Static analysis, Rigid segment method

1. INTRODUCTION

The trusses are geometrically unchanged shapes that consists of bars with constant cross section which are mutually connected in nodes.

They are used in supported structures of different purposes [1], like as halls, roof constructions, bridges, cranes, mono pole transmission line towers, etc. Massiveness of their use is justified by low load to weight ratio in comparison with the solid girder, less material consumption, it is easy to achieve uniformity by repeating the same basic elements, etc. On the other hand, the process of making trusses is much more complex compared to solid girder, which sometimes may call into question the justification of their application.

The trusses can be divided into two basic groups on planar and spatial trusses. In this paper, the analysis will be limited to the planar trusses. An example of a planar truss is shown in Figure 1. It consists of upper chord (1), lower chord (2), nodes (3), verticals (4) and diagonals (5).

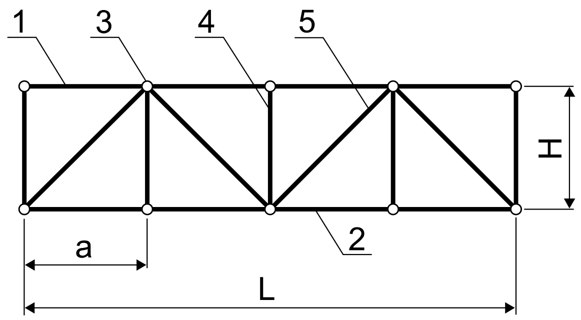


Figure 1: Planar truss

Plane trusses are loaded in bending as the whole, while the rods that make the truss are axially loaded, if it is assumed that the rods are hinged in the nodes rather than clamped [1]. In the real applications, the bars are usually clamped in the nodes, which leads to the bending loads in the bars, too. However, the bending deformations are negligiable small in relation to axial deformations of the bar. So, the bending deformations are not taken into account in static analysis.

This static analysis is an essential step in the truss design. For this purpose, the finite element method implemented in various available commercial software is the most recently used.

In this paper, the alternative approach, based on the rigid segment method [2] , will be presented. The discretization of the elastic bar will be made, so the bar elasticity was modelled by springs of the corresponding stiffness. The goal of this paper is to make an universal algorithm for static analysis of the planar truss.

2. THE DISCRETIZED MODEL OF THE PLANAR ELASTIC BAR

Figure 1(a) shows the discretized model of elastic bar which consist of two rigid segments of equal length $l_{jk}/2$, with the constant Young's modulus E, the constant cross section A and the constant cross section moment of inertia I_z.

The rigid segments are mutually connected with elastic joint element which allows the relative translation in axial and transverse direction (p_{jk} i q_{jk}), as well as the relative rotation around the axis perpendicular to motion plane $\left(r_{jk}\right)$.

Based on the reference [2], the stiffness of the springs in the joint element JE_{jk} should be obtained in the following manner:

$$c_{p_{jk}} = \frac{AE}{l_{jk}},\ c_{q_{jk}} = 12\frac{EI_z}{l_{jk}^{\ 3}},\ c_{r_{jk}} = \frac{EI_z}{l_{jk}}, \tag{1}$$

where:

$$l_{jk} = \sqrt{\left(x_k - x_j\right)^2 + \left(y_k - y_j\right)^2} \tag{2}$$

is the length of the *i*-th elastic bar that is determined based on coordinates x_j, y_j, x_k i y_k of points P_j i P_k in the initial equilibrium position in accordance to selected inertial coordinate system $Oxyz$.

*Corresponding author: Faculty of mechanical and civil engineering in Kraljevo, University of Kragujevac, nikolic.a@mfkv.kg.ac.rs

Since it is assumed here that the elastic rod is loaded only in the axial direction, then a discretized model of an elastic rod with only axial displacement p_{jk} can be used, while the displacements q_{jk} i r_{jk} can be ignored.

In the further, the proposed disretized model of the bar should be symbolic represented as in th figure 1(b).

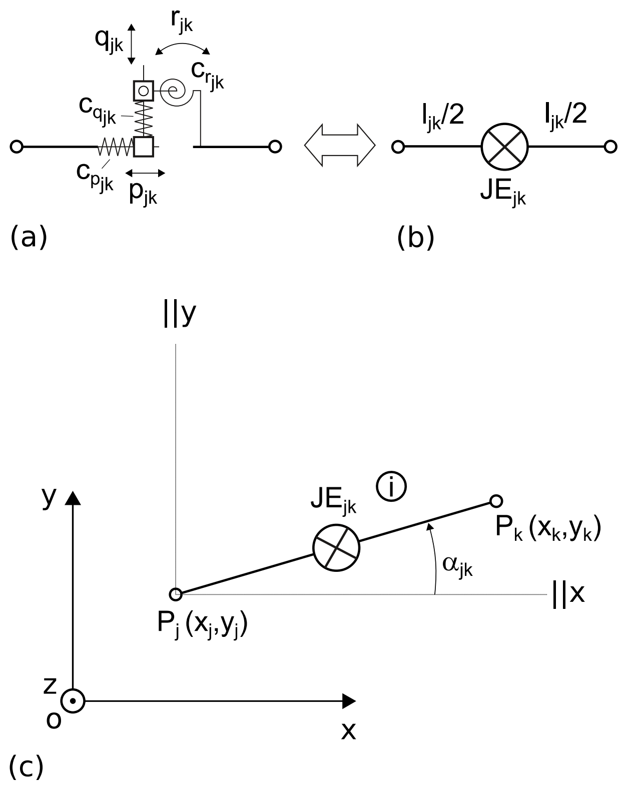


Figure 2: Discretization of the elastic bar into the rigid segments

3. THE POTENTIAL ENERGY OF THE PLANAR TRUSS

The potential energy of the planar truss should be obtained as the sum of the potential energies of the each rod that makes the truss in the following way:

$$\Pi = \frac{1}{2}\sum_{j=1}^{n}\sum_{k=1}^{n}\gamma_{jk}c_{p_{jk}}p_{jk}^{\ 2}, \tag{3}$$

where n is the total number of nodes in a truss, $\gamma_{jk}\left(j<k\right)$ represents an node connectivity identifier that has a value of 1 if the nodes are connected by a rod or a value of 0 if it is not the case, and p_{jk} represents a relative axial displacement of the rod that connects the nodes j and k and can be defined as follows:

$$p_{jk} = \left(\tilde{x}_{p_k} - \tilde{x}_{p_j}\right)\cos\alpha_{jk} + \left(\tilde{y}_{p_k} - \tilde{y}_{p_j}\right)\sin\alpha_{jk}, \tag{4}$$

where $\tilde{x}_{p_j}, \tilde{y}_{p_j}, \tilde{x}_{p_k}$ and $\tilde{y}_{p_k}$ are the horizontal and vertical displacements of points P_j i P_k in accordance to the initial equilibrium position, respectively, α_{jk} is the angle between the rod i and axis x in the initial equilibrium position, and it holds that:

$$\sin\alpha_{jk} = \frac{y_k - y_j}{l_{jk}},\ \cos\alpha_{jk} = \frac{x_k - x_j}{l_{jk}}. \tag{5}$$

If the terms (3) - (5) are taken into account now, the potential energy of a plane truss can be written as the positive definite square form in the following way:

$$\Pi = \frac{1}{2}\mathbf{v}^T\mathbf{K}\mathbf{v}, \tag{6}$$

where

$$\mathbf{v} = \left[v_1 \ldots v_j \ldots v_n\right]^T,$$

is the vector of truss node displacements in accordance to the initial equilibrium position whereas it holds that $v_{2i-1} = \tilde{x}_i$, $v_{2i} = \tilde{y}_i$, n represent the total number of truss nodes,

$$K = \begin{bmatrix} k_{1,1} & k_{1,2} & \cdots & k_{1,} & \cdots & k_{1,2n} \\ k_{2,1} & k_{2,2} & & k_{2,j} & & k_{2,2n} \\ \vdots & & \ddots & & & \vdots \\ k_{i,1} & k_{i,2} & \cdots & k_{i,j} & \cdots & k_{i,2n} \\ \vdots & & & & \ddots & \vdots \\ k_{2n,1} & k_{2n,2} & \cdots & k_{2n,j} & \cdots & k_{2n,2n} \end{bmatrix} \tag{7}$$

is the stiffness matrix of the considered planar truss where the stiffness coeficients should be obtained, based on the expression for potential energy (3), in the following manner:

$$k_{i,j} = \frac{\partial^2\Pi}{\partial v_i \partial v_j},\ \left(i,j=1,\ldots,n\right). \tag{8}$$

4. THE EQUILIBRIUM CONDITIONS OF A PLANAR TRUSS

Suppose that in the general case, the vectors of the external forces

$$\mathbf{F}_j = \left[F_{x,j}\ F_{y,j}\right]^T, \left(j=1,\ldots,n\right), \tag{9}$$

acts on the truss nodes, where $F_{x,j}$ and $F_{y,j}$ are the projections of the external force vector $\mathbf{F}_j$ on the coordinate axes x and y, respectively. It should be noted here that the external reaction forces, which derive from the boundary conditions, are also accounted in the above mentioned vector of external forces.

Now the total vector of external forces that acting on the truss can be written as:

$$\mathbf{F} = \left[\mathbf{F}_1^{\ T} \ldots \mathbf{F}_j^{\ T} \ldots \mathbf{F}_n^{\ T}\right]^T. \tag{10}$$

Due to the action of a predefined vector of external forces on a truss, it moves from the initial to the new equilibrium position.

By using the Lagrangian principle of virtual displacements [3], and on the basis of expressions (6) and (10), the condition that must be fulfilled into the new equilibrium position is:

$$-\mathbf{Kv}+\mathbf{F}=\mathbf{0}_{2n\times 1}. \tag{11}$$

By solving the previous system of equations by an unknown vector of the displacements in the truss nodes, it turns out that:

$$\mathbf{v}=\mathbf{K}^{-1}\mathbf{F}. \tag{12}$$

Before determining an unknown vector of displacements in truss nodes $\mathbf{v}$, it is necessary to define the boundary conditions. It is possible that some of the truss nodes are fixed as roller or hinged support types. In that case, the displacements in node j are limited, so it holds that $\tilde{x}_j=\tilde{y}_j=0$ in the case of hinged support, and $\tilde{x}_j=0$ or $\tilde{y}_j=0$, in the case of roller support, depending on the orientation of the roller, ie whether horizontally or vertically oriented. Therefore, it is necessary to remove the rows and columns in the stiffness matrix $\mathbf{K}$, and the rows in the vectors $\mathbf{v}$ and $\mathbf{F}$ which corresponds to the limited displacements.

After determining the vectors of unknown displacements of the nodes $\mathbf{v}$ from the expression (12), by returning the values of displacements for all nodes (including the zero values) into the expression (11), the unknown external reaction forces should be obtained.

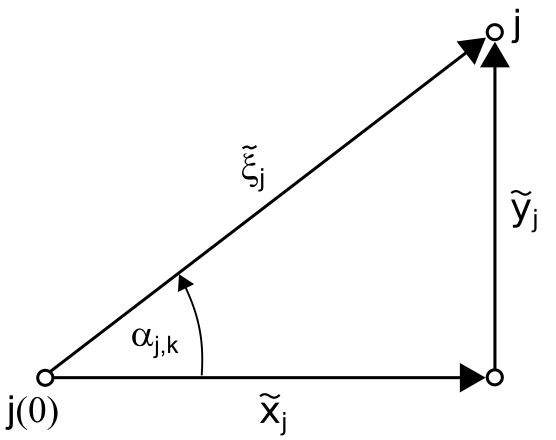


Figure 3: The displacements in the j-th truss node

Also, it is possible to determine the forces in all truss rods by using the previously determined node displacements, First, the axial deformations in nodes j and k, in the direction of bar which connect that nodes, should be defined as (see figure 3):

$$\tilde{\xi}_j=\tilde{x}_j\cos\alpha_{jk}+\tilde{y}_j\sin\alpha_{jk}, \tag{13}$$

$$\tilde{\xi}_k=\tilde{x}_k\cos\alpha_{jk}+\tilde{y}_k\sin\alpha_{jk}. \tag{14}$$

Based on the previous two expressions, the axial force which occurs in the rod that connects the nodes j and k may be defined as [4]:

$$N_{jk}=\frac{EA}{l_{jk}}\left[\left(\tilde{x}_k-\tilde{x}_j\right)\cos\alpha_{jk}+\left(\tilde{y}_k-\tilde{y}_j\right)\sin\alpha_{jk}\right]. \tag{15}$$

Now, the normal stress (pressure or tension) in each truss rod can be defined as follows [4]:

$$\sigma_{jk}=\frac{N_{jk}}{A}. \tag{16}$$

5. NUMERICAL EXAMPLES

5.1. Example 1

Figure 4 shows the system of two elastic rods of length $\sqrt{2}a$ with the constant values of cross section area A and cross section moment of inertia around the perpendicular axis I_z, which are mutually connected in node 2. The bar material has the Young modulus E. The nodes 1 and 3 are fixed with hinged support. There is a constant horizontal external force in the node 2. It is necessary to determine the displacements in the node 2, as well as the reaction forces in the hinged supports 1 and 3.

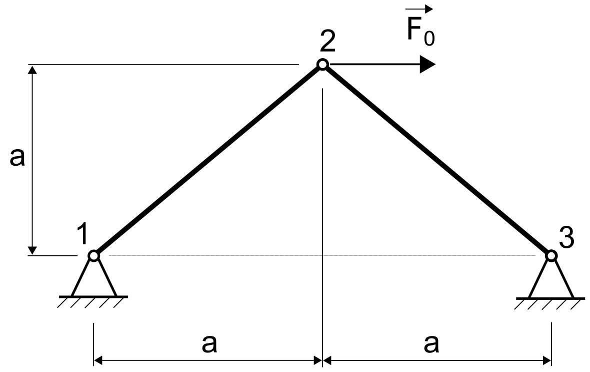


Figure 4: Truss structure with the two rods

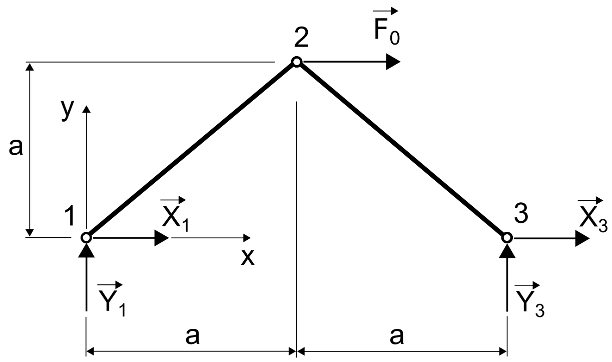


Figure 5: The free truss structure with the two rods

The coordinates of truss nodes are $x_1=y_1=0$, $x_2=y_2=a$, $x_3=2a$, $y_3=0$ and node connectivity identifiers are $\gamma_{12}=\gamma_{23}=1$, $\gamma_{13}=\gamma_{31}=\gamma_{21}=\gamma_{32}=0$. By using the given parameters and the expressions (1) to (8) the stiffness matrix of the considered truss reads:

$$\mathbf{K}=\frac{\sqrt{2}}{4}\frac{AE}{a}\begin{bmatrix}1 & 1 & -1 & -1 & 0 & 0\\ 1 & 1 & -1 & -1 & 0 & 0\\ -1 & -1 & 2 & 0 & -1 & 1\\ -1 & -1 & 0 & 2 & 1 & -1\\ 0 & 0 & -1 & 1 & 1 & -1\\ 0 & 0 & 1 & -1 & -1 & 1\end{bmatrix}. \tag{17}$$

Based on the expression (10), the vector of the external forces that acts on the considered truss reads:

$$\mathbf{F}=\begin{bmatrix}X_1 & Y_1 & F_0 & 0 & X_3 & Y_3\end{bmatrix}^T. \tag{18}$$

Also, the vector of the displacements in the truss nodes has the following form:

$$\mathbf{v}=\begin{bmatrix}\tilde{x}_1 & \tilde{y}_1 & \tilde{x}_2 & \tilde{y}_2 & \tilde{x}_3 & \tilde{y}_3\end{bmatrix}^T. \tag{19}$$

If the boundary conditions are now applied, then there is no displacements in nodes 1 and 3, ie it holds that $\tilde{x}_1=\tilde{y}_1=\tilde{x}_3=\tilde{y}_3=0$. Now, based on the expression (12), the vector of displacement of the truss nodes should be obtained as:

$$\mathbf{v}=\begin{Bmatrix}\tilde{x}_2\\ \tilde{y}_2\end{Bmatrix}=\frac{4a}{\sqrt{2}AE}\begin{bmatrix}2 & 0\\ 0 & 2\end{bmatrix}^{-1}\begin{Bmatrix}F_0\\ 0\end{Bmatrix}=\begin{Bmatrix}\sqrt{2}\dfrac{aF_0}{AE}\\ 0\end{Bmatrix}. \tag{20}$$

If the obtained values of nodes displacements are now returned to expression (11), it turns out that:

$$\begin{Bmatrix}X_1\\ Y_1\\ F_0\\ 0\\ X_3\\ Y_3\end{Bmatrix}=-\frac{\sqrt{2}}{4}\frac{AE}{a}\begin{bmatrix}1 & 1 & -1 & -1 & 0 & 0\\ 1 & 1 & -1 & -1 & 0 & 0\\ -1 & -1 & 2 & 0 & -1 & 1\\ -1 & -1 & 0 & 2 & 1 & -1\\ 0 & 0 & -1 & 1 & 1 & -1\\ 0 & 0 & 1 & -1 & -1 & 1\end{bmatrix}\begin{Bmatrix}0\\ 0\\ \sqrt{2}\dfrac{aF_0}{AE}\\ 0\\ 0\\ 0\end{Bmatrix},$$

ie,

$$\begin{Bmatrix}X_1\\ Y_1\\ F_0\\ 0\\ X_3\\ Y_3\end{Bmatrix}=\begin{Bmatrix}-0.5F_0\\ -0.5F_0\\ F_0\\ 0\\ -0.5F\\ 0.5F\end{Bmatrix}. \tag{21}$$

Since the nodes displacements are determined in the expression (20), now the axial forces as well as the stresses in the rods can be determined from the expression (15) as:

$$N_{12}=-N_{23}=\frac{F_0}{\sqrt{2}},\ \sigma_{12}=-\sigma_{23}=\frac{F_0}{\sqrt{2}A}. \tag{22}$$

The obtained results for nodes displacements (20), the external force vector (21), as well as the axial forces and stresses in the rods (22) fully match the results from the reference [4], where the static analysis was performed by the finite element method. The reason for this matching is because the axial stiffness value of the elastic rod is the same in both methods (rigid segments method and finite element method).

5.2. Example 2

The static analysis of the truss showed in the figure 6 should be performed in this example. The considered truss consist of nine rods that are interconnected by six nodes. In nodes 2, 4, and 6, the vertical forces of intensities 50, 100 and 50 [kN] are acting vertically downwards, respectively. The truss is connected with a ground in nodes 1 and 5, with a hinged and roller support, respectively. The free truss is shown in figure 7.

The length of the bars in truss is 3 [m], the cross section area of the bars is $A=3\cdot10^{-4}\,[\mathrm{m}^2]$ and the Young's modulus of bars material is $E=0.7\cdot10^{11}\,[\mathrm{Pa}]$.

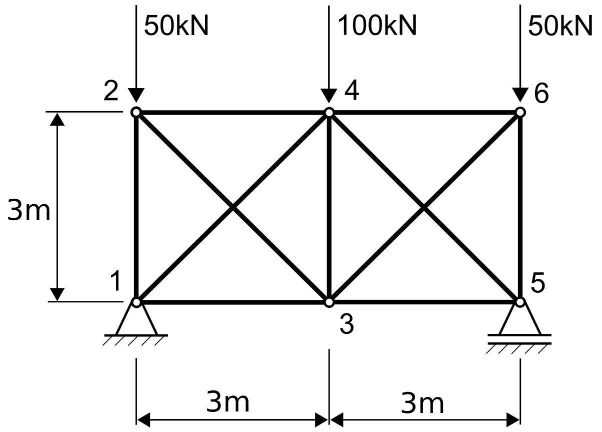


Figure 6: The truss

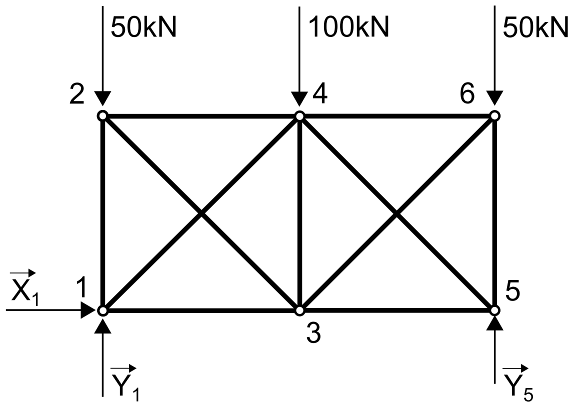


Figure 7: The free truss

The coordinates of truss nodes are $x_1=x_2=0$, $x_3=x_4=3[\mathrm{m}]$, $x_5=x_6=6\,[\mathrm{m}]$, $y_1=y_3=y_5=0$, $y_2=y_4=y_6=3[\mathrm{m}]$ and node connectivity identifiers are $\gamma_{12}=\gamma_{13}=\gamma_{14}=1$, $\gamma_{23}=1, \gamma_{24}=1$, $\gamma_{34}=\gamma_{35}=\gamma_{36}=1$, $\gamma_{45}=1, \gamma_{46}=1$ and $\gamma_{56}=1$. All other node connectivity identifiers are equal to zero. By using the boundary conditions it holds that $\tilde{x}_1=\tilde{y}_1=0$ i $\tilde{y}_5=0$.

The data for displacements in all truss nodes are shown in Table 1. The obtained displacement values are identical to the values from reference [5] where the finite element method is used, with assuming that the rods are hinged in the nodes. Also, the same table shows the node displacement values obtained by using the software package SAP2000 [6].

By comparing the obtained results it can be noticed that the difference between the results is negligible and does not exceed 0.35%. The reason for the occurrence of this difference is that in the software package SAP2000 was assumed that the rods are hinged in the nodes, as assumed in this paper and in the reference [5], rather than clamped.

Also, Table 2 shows the axial forces in the rods obtained by applying the proposed approach and the software package SAP 2000. And here the difference in the obtained values is negligible, the maximum is 0.64%. A better insight into the axial force distribution in the rods can be obtained from the diagram shown in figure 8.A slight difference in the obtained results leads us to the conclusion that the using of a assumption that the rods are hinged in the nodes is justified because it significantly simplifies the analysis and negligible affects the accuracy.

Table 1: Displacements in the truss nodes

Node	Displacements $\tilde{x}_i \cdot 10^3$ [m] MKS	Displacements $\tilde{x}_i \cdot 10^3$ [m] MKE	Displacements $\tilde{y}_i \cdot 10^3$ [m] MKS	Displacements $\tilde{y}_i \cdot 10^3$ [m] MKE
1	0 -	0	0 -	0
2	7.1429 0.11%	7.1353	-9.0386 0.22%	-9.0187
3	5.2471 0.18%	5.2375	-16.2965 0.13%	-16.2761
4	5.2471 0.18%	5.2375	-20.0881 0.23%	-20.0426
5	10.4942 0.18%	10.475 0	0 -	0
6	3.3513 0.35%	3.3396	-9.0386 0.22%	-9.0187

MKS – Rigid segment method
MKE – Finite element method (SAP2000)

Table 2: Axial forces in rods

N_{jk} [kN]	MKS	MKE
N_{12}	-63.27 0.36%	-63.5
$N_{13} = N_{35}$	36.73 0.35%	36.86
$N_{14} = N_{45}$	-51.94 0.36%	-52.13
$N_{23} = N_{36}$	18.77 0.64%	18.89
$N_{24} = N_{46}$	-13.27 0.67%	-13.36
N_{34}	-26.54 -	-26.54

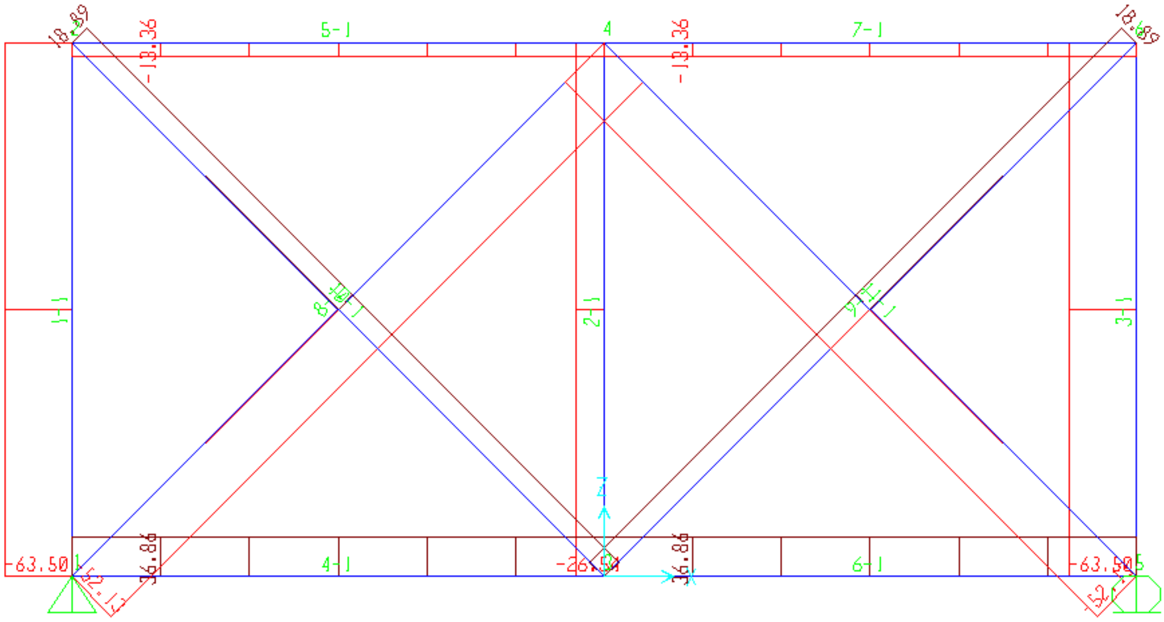


Figure 8: Diagram of axial forces in truss nodes (SAP2000)

6. CONCLUSION

This paper presents a general approach for the static analysis of planar truss. The equilibrium conditions of a planar truss were derived by using the basic principles of analytical mechanics. The efficiency and accuracy of the proposed approach was demonstrated through the two numerical examples. The obtained results are identical to the results obtained by using finite element methods. The proposed algorithm can be easily implemented by using even free programming languages, e.g. Python, R, Octave, Scilab, Maxima, etc.

The proposed method of discretizing the elastic rod in the plane can also be used for the analysis of planar frame structures. In that case, joint element with three degrees of freedom, which was proposed at the beginning of work, would be used. The analysis of the planar frames will be the subject of further research by the authors.

ACKNOWLEDGMENTS

This research was supported by the Ministry of Education, Science and Technological Development of the Republic of Serbia (Grant Nos. ON174016 and TR35038). This support is gratefully acknowledged.

REFERENCES

[1] Z. Petković, D. Ostrić, “Metalne konstrukcije u mašinogradnji 1”, Mašinski fakultet u Beogradu, Beograd, (1996)

[2] A. Nikolić, S. Šalinić, “Dynamics of the Rotating Cantilever Beam”, Proceedings of IX International Conference “Heavy Machinery HM 2017”, Zlatibor (Serbia), June 28-July 1, pp.D(7-12), (2017)

[3] A. I. Lurie, “Analytical Mechanics”, Springer Science & Business Media, (2013)

[4] A. Öchsner, R. Makvandi, "Finite Elements for Truss and Frame Structures: An Introduction Based on the Computer Algebra System Maxima", Springer, (2018).

[5] A. J. FERREIRA, "*MATLAB codes for finite element analysis: solids and structures",* Springer Science & Business Media, (2008)

[6] https://www.csiamerica.com/products/sap2000