NON-LINEAR GEOMETRIC ANALYSIS OF DEFLECTIONS IN METALLIC TRUSSES

Sebastião L. Pimenta
Leonardo M. Costa
sebapimenta@gmail.com
leonardom.costa@yahoo.com.br
Department of Civil Engineering - CCTS, Paraíba State University
Av. Cel. Pedro Targino, s / n, 58233-000 Araruna, PB, Brazil
Daniel C. Silva
daniel.costa.silva@hotmail.com
Postgraduate Program in Civil Engineering, State University of Rio de Janeiro
Sao Francisco Xavier Street, 524, Maracanã, 20550-900 Rio de Janeiro, RJ, Brazil
Francisco R. R. Sousa
ramon_desousa@hotmail.com
Department of Civil Engineering - CCTS, Paraíba State University
Av. Cel. Pedro Targino, s / n, 58233-000 Araruna, PB, Brazil
José R. L. Paixão Filho
joserobertopaixao@hotmail.com
Department of Civil Engineering - CCTS, Paraíba State University
Av. Cel. Pedro Targino, s / n, 58233-000 Araruna, PB, Brazil
Luiz R. S. Linhares
luizricardopb@gmail.com
Postgraduate Program in Civil Engineering, Federal University of Rio Grande do Norte
Senador Salgado Filho Avenue, 3000, Lagoa Nova University Campus, 59078-970 Natal, RN, Brazil

Abstract. Geometric non-linearity consists of determining the stresses and displacements of a structure considering the deformed state of the structure after load application, and therefore the equilibrium equations must be reformulated for each of these changes. The present paper, in this sense, has proposed to elaborate a software capable of analyzing the deflections of a structure of the trellis type, implementing the changes of geometry with each applied load step. In order to perform the calculations, the Finite Element Method was used, and the algorithm was written in the Python programming language. With the results obtained in the software developed, and through comparisons with SAP2000 V15, it was verified that the simplified approach of geometric non-linearity, which was implemented in the computational program elaborated, is very close to the results of SAP2000 as the load increase decreases. The results show that the lattice node analyzed with the greatest divergence between the software had a decrease in the difference between the deflections of the two software from 5.61% to 1.89% when changing the number of load steps, from two to one hundred steps.

Keywords: Nonlinear Analysis, Structural analysis, Metallic Trusses, Computational Program.

1 Introduction

Metal trusses are a recurring solution as cover elements. Factors such as its high strength-to-weight ratio, in addition to its aesthetic appeal, are crucial for its use, especially in buildings with large free spans. According to Lacerda [1], trusses can be defined as a stable arrangement of interconnected thin bars, where the connections are articulated and the bars are connected by frictionless pins, so that no moment can be transmitted by this connection. This is the definition of an ideal framework, but the actual structures shy away from this premise, either by connection design settings or by the effect of time. Structurally, the stresses from joints with more than one pin and / or friction are small, and not determinant in structural sizing. Thus, the trusses are characterized by being an arrangement of bars that only transmit axial forces, except in cases where the bars are subjected to transverse forces, causing the generation of shear forces and bending moments along the elements.

Within this context, the analysis of truss arrows is necessary to ensure their stability and meet the requirements regarding the Limit of Service (ELS) standards set forth in standards such as ABNT NBR 8800/2008 [2]. However, the calculation methods used in linear analysis of structures may not be satisfactory in certain situations, since these methods do not include the study of physical and / or geometric nonlinearity of structural components.

Cunha [3], points out that due to the influence of the rigidity of the structural element in its operation, nonlinear analysis has greater relevance in slender structures, which are being used more frequently due to economic and architectural factors. Thus, greater knowledge about the physical and geometric nonlinearity is required by the designer.

However, the main consequence of a nonlinear analysis is the increased complexity of the problem, making it impossible to perform an analytical solution. In such cases, the use of computational tools and numerical approximations becomes practically mandatory.

Thus, the present work aims to present a software capable of calculating lattice arrows taking into account the geometric nonlinearity, using the Finite Element Method formulated from the displacement method. Another prerogative of this work is to compare the values obtained in the linear and nonlinear analysis in a flat truss.

2 Geometric Nonlinearity

Lacerda [1] explains that when a structure undergoes large deformations, the equilibrium equations valid for its undisturbed state become incoherent. In the deformed state, equilibrium equations need to be reformulated with each change in structural geometry, which causes loss of linearity in the displacement and deformation relations. This type of nonlinearity is called geometric nonlinearity.

According to Benjamin [4], geometric nonlinearity is relevant in cases where due to the magnitude of the displacements arises the need to write equilibrium equations in relation to the deformed configuration of the structure.

The performance of nonlinear analyzes may be more important on certain occasions, as Pinto and Ramalho [5] say, in the design of tall buildings one must pay attention to the problem of geometric nonlinearity when the structure is requested simultaneously by vertical loads and horizontal. For, the vertical loading acting on the displaced structure may cause the manifestation of additional efforts capable of leading it to collapse, since the structure was designed taking into account its initial geometry. Such effects can be neglected if the structure is rigid enough, but in flexible structures these effects are quite significant and must be considered. Thus, structures can be classified into moving node structures or fixed node structures, according to the importance of second-order effects in the analysis.

3 Matrix analysis

For Vaz [6], matrix analysis is a way to systematize mathematical operations in the study of structural behavior, using vectors and matrices. For this, this systematization is based on the idea of local and global coordinate system. From this prerogative, it becomes possible to establish element stiffness matrices in the local and global systems, as well as vectors of forces and nodal displacements in the mentioned systems.

Due to the unique characteristics of the trusses (purely axial forces), the nodal displacements of the structure can be taken as the unknowns of the equilibrium equations. In this way the number of free displacements is called the degree of freedom of the structure. The flat truss structure analyzed in this paper has two degrees of freedom per node. Each node is associated with a displacement (d_i) towards the local x axis. The angle θ defines the rotation of the axis of the bar in relation to the global system, which allows to express the displacement in the global axes. Such a configuration can be better understood from Fig. 1.

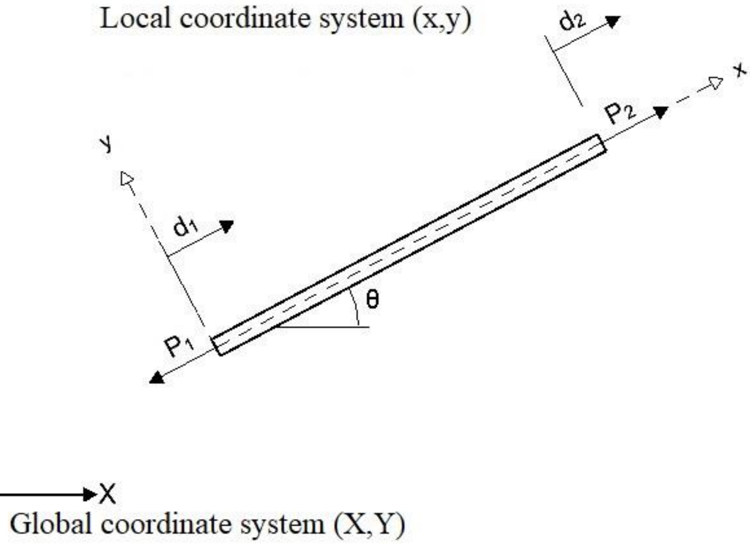


Figure 1. Framework Bar Coordinate Systems

From Fig. 1, it is noted that for the bar to be in equilibrium it is necessary that:

$$P_1 = -\ A\sigma \quad (1)$$

$$P_2 = A\sigma \quad (2)$$

Where "A" is the cross-sectional area of the bar and is the stress acting on it. For the sake of simplicity, Hooke's law can be applied to the calculation of arrows, provided that the component material of the truss elements has linear stress-strain behavior. So for each bar:

$$\sigma_i = E\varepsilon_i \quad (3)$$

Where E represents the modulus of elasticity of the material. To relate the elongations (or shortenings) of the bars (δ_i) to their longitudinal deformations (ε_i) we use the following expression:

$$\varepsilon_i = \frac{\sigma_i}{L_i} \quad (4)$$

Substituting, respectively, Eq. (3) and (4) in Eq. (1) and (2), we obtain:

$$P_1 = -\ EA\frac{d_2 - d_1}{L} \quad (5)$$

$$P_2 = EA\frac{d_2 - d_1}{L} \quad (6)$$

In matrix form:

$$\frac{EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} d_1 \\ d_2 \end{Bmatrix} = \begin{Bmatrix} P_1 \\ P_2 \end{Bmatrix}. \quad (7)$$

The displacements and forces can be decomposed in the global coordinate system X and Y using the following equations:

$$d_i = d_{iX}\cos\theta + d_{iY}\operatorname{sen}\theta. \quad (8)$$

$$P_{iX} = P_i\cos\theta. \quad (9)$$

$$P_{iY} = P_i\operatorname{sen}\theta. \quad (10)$$

In matrix from we have:

$$\begin{bmatrix} \cos\theta & \operatorname{sen}\theta & 0 & 0 \\ 0 & 0 & \cos\theta & \operatorname{sen}\theta \end{bmatrix}\begin{Bmatrix} P_1 \\ P_2 \end{Bmatrix} = \begin{Bmatrix} P_{1X} \\ P_{1Y} \\ P_{2X} \\ P_{2Y} \end{Bmatrix}. \quad (11)$$

$$\begin{bmatrix} \cos\theta & 0 \\ \operatorname{sen}\theta & 0 \\ 0 & \cos\theta \\ 0 & \operatorname{sen}\theta \end{bmatrix}\begin{Bmatrix} P_1 \\ P_2 \end{Bmatrix} = \begin{Bmatrix} P_{1X} \\ P_{1Y} \\ P_{2X} \\ P_{2Y} \end{Bmatrix}. \quad (12)$$

Substituting Eq. (11) and (12) for Eq. (7) gives the following configuration:

$$\begin{bmatrix} \cos\theta & 0 \\ \operatorname{sen}\theta & 0 \\ 0 & \cos\theta \\ 0 & \operatorname{sen}\theta \end{bmatrix}\frac{EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{bmatrix} \cos\theta & \operatorname{sen}\theta & 0 & 0 \\ 0 & 0 & \cos\theta & \operatorname{sen}\theta \end{bmatrix}\begin{Bmatrix} d_{1X} \\ d_{1Y} \\ d_{2X} \\ d_{2Y} \end{Bmatrix} = \begin{Bmatrix} P_{1X} \\ P_{1Y} \\ P_{2X} \\ P_{2Y} \end{Bmatrix}. \quad (13)$$

$$\frac{EA}{L}\begin{bmatrix} \cos^2\theta & \cos\theta\operatorname{sen}\theta & -\cos^2\theta & -\cos\theta\operatorname{sen}\theta \\ \cos\theta\operatorname{sen}\theta & \operatorname{sen}^2\theta & -\cos\theta\operatorname{sen}\theta & -\operatorname{sen}^2\theta \\ -\cos^2\theta & -\cos\theta\operatorname{sen}\theta & \cos^2\theta & \cos\theta\operatorname{sen}\theta \\ -\cos\theta\operatorname{sen}\theta & -\operatorname{sen}^2\theta & \cos\theta\operatorname{sen}\theta & \operatorname{sen}^2\theta \end{bmatrix}\begin{Bmatrix} d_{1X} \\ d_{1Y} \\ d_{2X} \\ d_{2Y} \end{Bmatrix} = \begin{Bmatrix} P_{1X} \\ P_{1Y} \\ P_{2X} \\ P_{2Y} \end{Bmatrix}. \quad (14)$$

Equation (14) is the one that will be implemented in the arrow calculation system using the Finite Element Method.

4 Methodology

To fulfill the proposed objectives for this work it was necessary to create a computational routine developed in Python, besides the use of the structural analysis software SAP2000 v.15.

Python's standard Integrated DeverLopment Environment (IDLE) was used to develop the graphical, logical, and operational interface of software capable of calculating arrows in flat trusses using the concepts and fundamentals of Finite Element Methods. This computational tool was developed aiming at the construction of a simple and intuitive graphical interface to provide a better user adaptability to the platform.

The proposed software was named “SATE” (Step Truss Analysis Software). It is noteworthy that SATE is limited only to calculating truss arrows with nodal loads, so it does not take into account own weight or any other load outside the nodes. This software has an user-friendly and simplified interface, with a grid that facilitates the design of the structure, and is aesthetically and functionally very similar to other software of this type. The figure below shows the SATE interface.

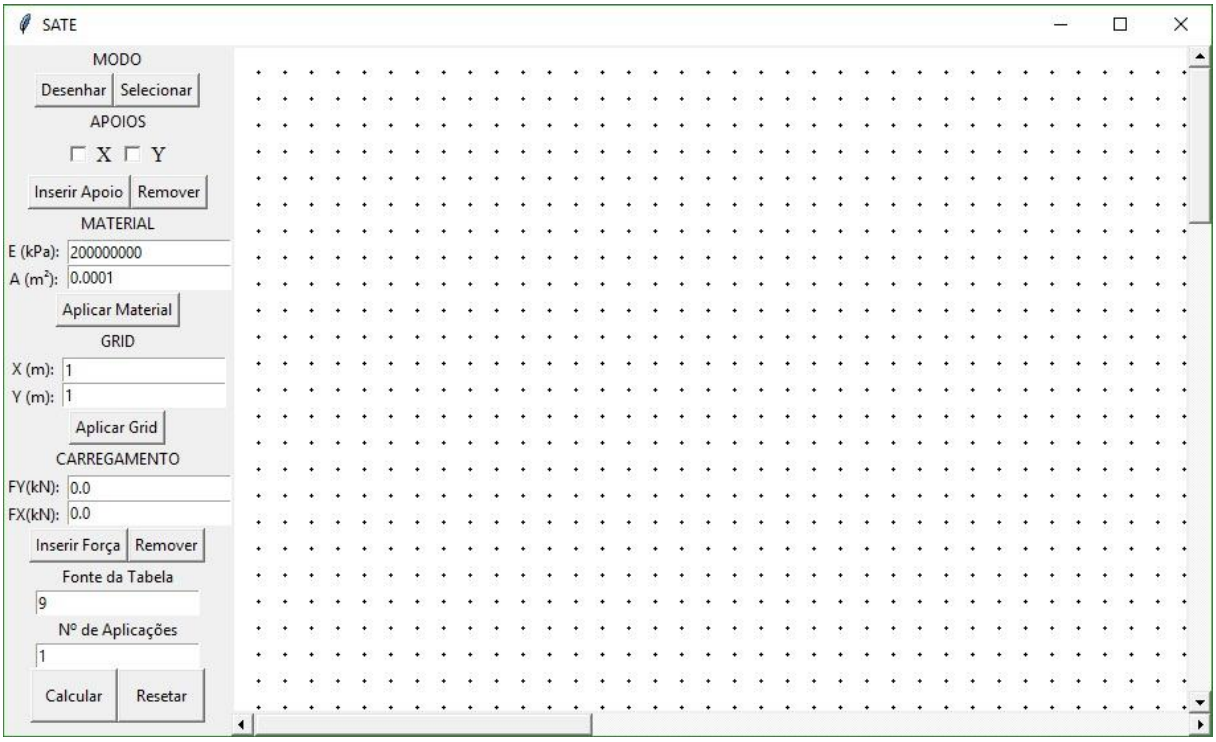


Figure 2. SATE's interface

In the software interface there are five main sections and two other configuration fields, the five main sections are: MODO, APOIOS, MATERIAL, GRID and CARREGAMENTO. In the MODO section there is the draw mode, which will allow the user to create the truss bars by simply clicking on grid points, and also the select mode which will allow the user to select bars and nodes in order to delete them, insert supports or insert forces. In the APOIOS section you can define the movement restriction that the selected point will have. In the MATERIAL section you can define the modulus of elasticity and the area of the bars, it is important to emphasize that all bars will have equal area and modulus of elasticity. In the GRID section it is possible to define the distance between grid points on both the X and Y axis. In the CARREGAMENTO section you can apply forces on the X and Y axes of the selected node, forces can only be applied to nodes. Apart from the five main sections there are also two other fields that are "Fonte da Tabela", where you can define the font size that the table generated in the results will have, and the "Nº de Aplicações" field which will define how many steps the load will be applied, where one application will be the same as a geometric linear analysis and from two applications the geometric nonlinear analysis will start.

After modeling the structure in SATE, the output of results is in the form of graphs and tables, where for each application of a portion of the force is drawn a line corresponding to the deformation of the structure. With the results obtained by SATE, the following comparative tests were performed:

- Comparison between arrows obtained and arrows calculated by linear analysis using SAP2000 v.15;
- Arrows calculated using geometric nonlinear analysis using SATE and arrows calculated using geometric nonlinear analysis (P-Delta) using SAP2000;
- Trusses analyzed linearly and trusses analyzed with geometric nonlinearity.

The comparative analysis was performed using the Howe truss presented in Fig. 3. This truss is a theoretical model formed by steel bars with cross-sectional area equal to 1.00 cm². An elastic modulus of 200 GPa was adopted for steel.

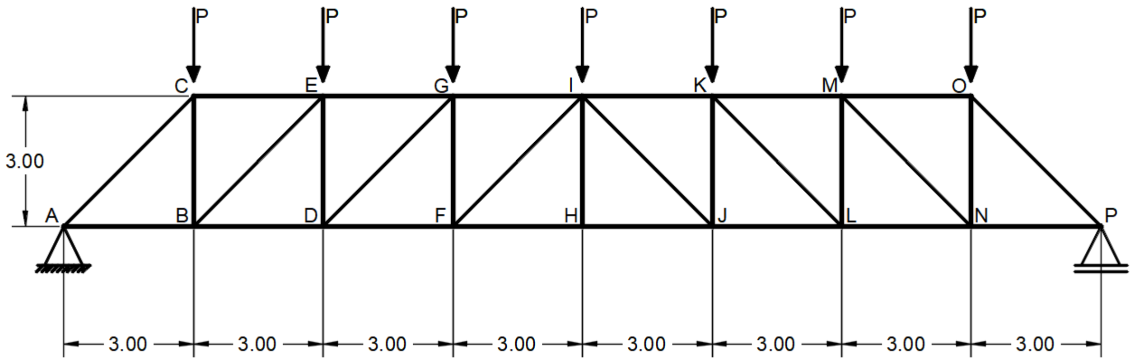


Figure 3. Howe truss configuration analyzed (Dimensions in meters)

For each of the mentioned comparative analyzes three different P loads were used, one of 3.00 kN, one of 10.00 kN and another of 100.00 kN. The load of 3.00 kN was chosen because the arrow caused by it is within the limits of L / 350, established by ABNT NBR 8800/2008 [2], this load represents a load closer to a real load. The loads of 10.00 and 100.00 kN were chosen to analyze the theoretical behavior of the truss when subjected to large loads.

Once the characteristics of the base truss were defined, it was first analyzed linearly in SAP2000 v.15 to obtain arrows, and submitted to the calculation routines programmed in the developed software, in order to make the comparison between the two software through percentage error between the methods.

To perform the second comparative analysis, the values obtained via SATE were first exposed, applying the force divided in 2 and 100 times portions for each load. The variation in the number of portions aims to visualize the behavior of the structure as the number of applications increases. Then the results obtained with the nonlinear geometric analysis of SAP2000 v.15 were displayed. Finally, a comparison was made between the two methods to verify the effectiveness of this type of analysis.

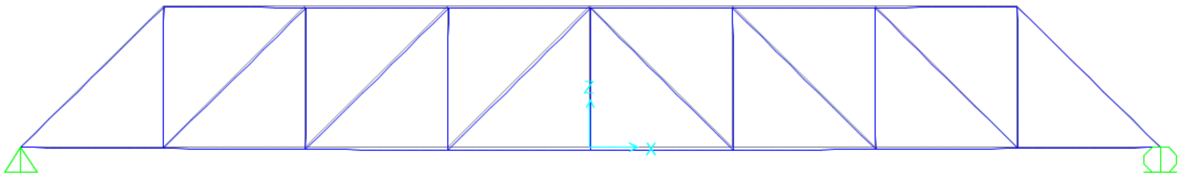
The nonlinear geometric analysis of the truss performed in the proposed software is achieved in load increments, in other words, the force before being applied will be divided into n portions, so that with each application of a portion of the force the structure will suffer changes due to deformations before the application of the next portion. In order to define the number of increments in which the force will be applied, a field has been placed in the developed program where the user can define how many times the force will be divided.

The third comparative analysis is the analysis between the linearly obtained arrows and the geometric nonlinearly obtained arrows. This analysis aims to verify how the geometric nonlinearity can influence the values of the arrows. In this analysis the values of each software were analyzed separately, that is, the linear arrows of SAP2000 v.15 were compared with nonlinear arrows also of SAP2000 v.15, in the same way for the developed software.

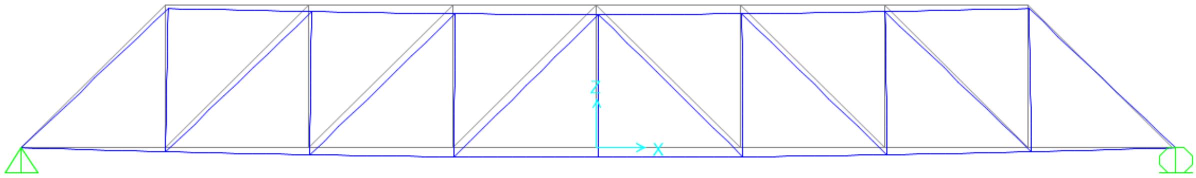
5 Results

5.1 Efficiency of SATE linear analysis

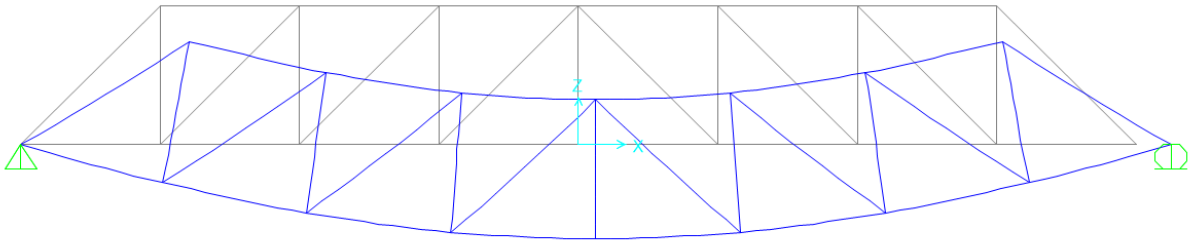
The comparative efficiency study between the already established SAP2000 v.15 and SATE was started with the linear analysis of the Howe truss presented in Fig. 3 for the loads of 3.00, 10.00 and 100.00 kN. The resulting arrows obtained in SAP 200 v.15 for each of the loads cited are shown in the Fig. 4.



(a)



(b)



(c)

Figure 4. Deformed structure in SAP2000 to P = 3.00 kN (a), P = 10.00 kN (b) and P = 100.00 kN (c)

The modeling of the structure in SATE can be verified through Fig. 5, for the load of 10.00 kN. It is noteworthy that in this step, linear analysis is obtained by applying the entire load at once, that is, without dividing the force. For loads of 3.00 kN and 100.00 kN the modeling is done analogously, replacing only the loads.

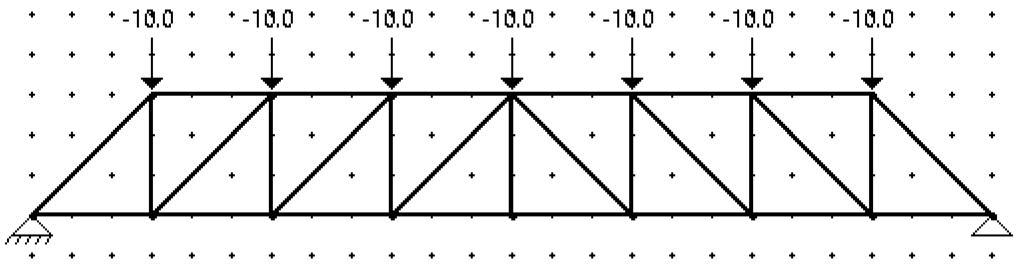
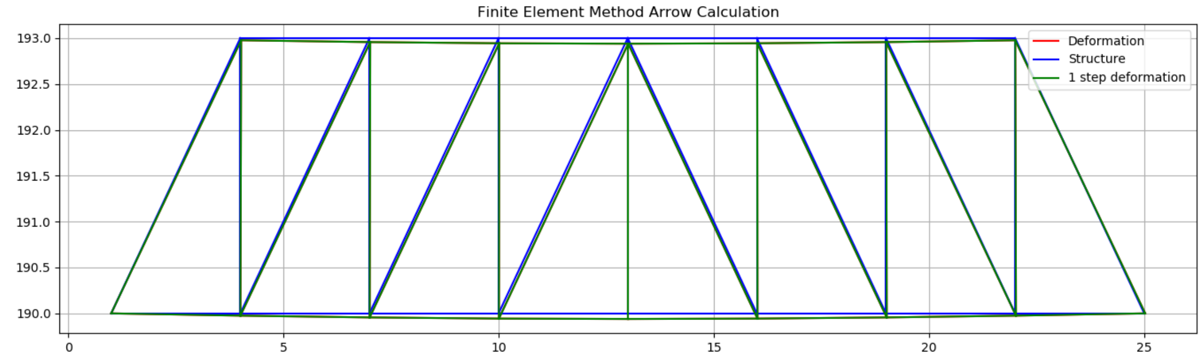
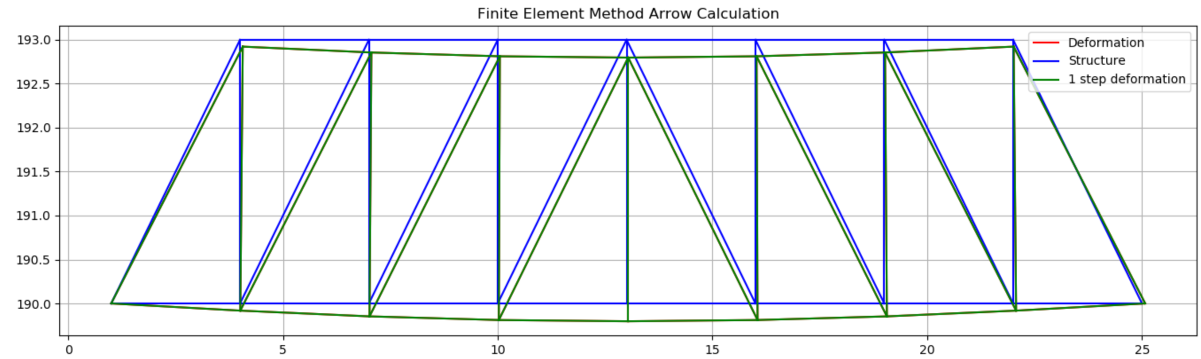


Figure 5. Howe truss modeling on SATE with 10.00 kN load

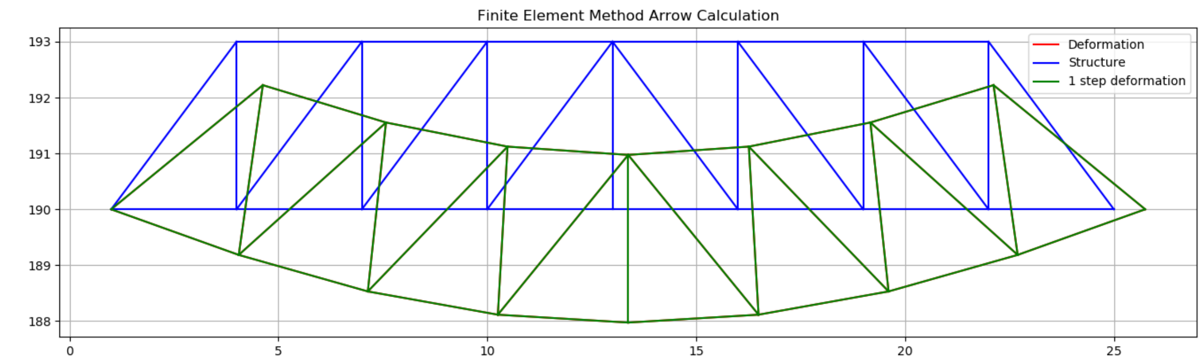
Once the lattice was modeled and the characteristics of the bars were defined, the linear analysis was performed using the SATE that resulted in the exposed by Fig. 6.



(a)



(b)



(c)

Figure 6. Deformed from the structure for P = 3.00 kN (a), P = 10.00 kN (b) and P = 100.00 kN (c)

Numerically, the nodal displacements obtained in both software are presented in Tables 1, 2 and 3 + 2 as a function of the applied load. These tables also include the percentage difference between SATE and SAP2000 v.15 results.

Table 1. Arrows obtained for P=3.00 kN

Node	**Arrows obtained (m)**				**Difference (%)**	
	SAP2000		**SATE**			
	Direction X	Direction Y	Direction X	Direction Y	Direction X	Direction Y
A	0.0000	0.0000	0.0000	0.0000	-	-
B	0.0016	-0.0245	0.0016	-0.0245	0.00	0.00
C	00189	-0.0234	0.0189	-0.0234	0.00	0.00
D	0.0043	-0.0441	0.0043	-0.0441	0.00	0.00
E	0.0173	-0.0434	0.0173	-0.0434	0.00	0.00

F	0.0077	-0.0566	0.0076	-0.0566	-1.30	0.00
G	0.0146	-0.0564	0.0146	-0.0563	0.00	-0.18
H	0.0113	-0.0608	0.0113	-0.0608	0.00	0.00
I	0.0113	-0.0608	0.0113	-0.0608	0.00	0.00
J	0.0149	-0.0566	0.0148	-0.0566	-0.67	0.00
K	0.0079	-0.0564	0.0079	-0.0563	0.00	-0.18
L	0.0182	-0.0441	0.0182	-0.0441	0.00	0.00
M	0.0052	-0.0434	0.0052	-0.0434	0.00	0.00
N	0.0209	-0.0245	0.0209	-0.0245	0.00	0.00
O	0.0036	-0.0234	0.0036	-0.0234	0.00	0.00
P	0.0225	0.0000	0.0225	0.0000	0.00	-

Table 2. Percentage difference between arrows obtained for P = 10.00 kN

Node	Arrows obtained (m)				Difference (%)	
	SAP2000		SATE			
	Direction X	Direction Y	Direction X	Direction X	Direction Y	Direction Y
A	0,0000	0,0000	0,0000	0,0000	-	-
B	0,0053	-0,0816	0,0053	-0,0816	0,00	0,00
C	0,0630	-0,0779	0,0630	-0,0778	0,00	-0,13
D	0,0143	-0,1470	0,0142	-0,1470	-0,70	0,00
E	0,0578	-0,1447	0,0578	-0,1447	0,00	0,00
F	0,0255	-0,1886	0,0255	-0,1886	0,00	0,00
G	0,0488	-0,1879	0,0488	-0,1878	0,00	-0,05
H	0,0375	-0,2027	0,0375	-0,2027	0,00	0,00
I	0,0375	-0,2027	0,0375	-0,2027	0,00	0,00
J	0,0495	-0,1886	0,0495	-0,1886	0,00	0,00
K	0,0263	-0,1879	0,0263	-0,1878	0,00	-0,05
L	0,0608	-0,1470	0,0607	-0,1470	-0,16	0,00
M	0,0173	-0,1447	0,0173	-0,1447	0,00	0,00
N	0,0689	-0,0816	0,0697	-0,0816	1,15	0,00
O	0,0120	-0,0779	0,0120	-0,0778	0,00	-0,13
P	0,0750	0,0000	0,0750	0,0000	0,00	-

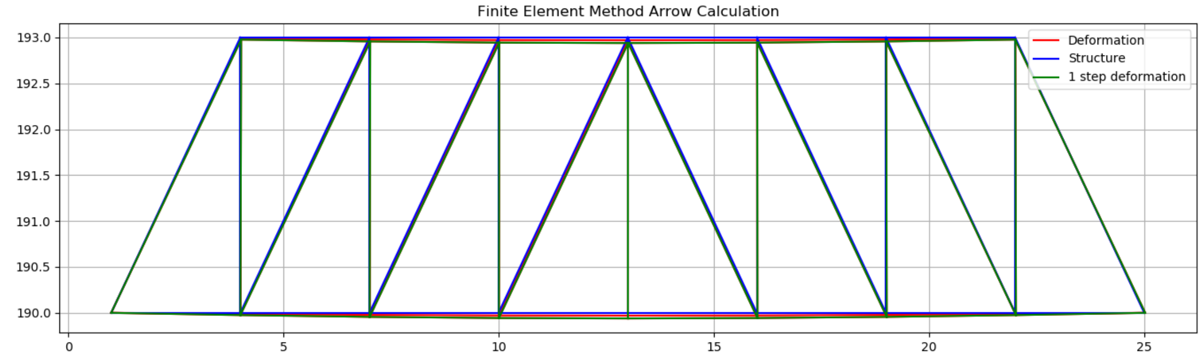
Table 3. Percentage difference between arrows obtained for P = 100.00 kN

Node	Arrows obtained (m)				Difference (%)	
	SAP2000		SATE			
	Direction X	Direction Y	Direção X	Direction X	Direction Y	Direção Y
A	0,0000	0,0000	0,0000	0,0000	-	-
B	0,0525	-0,8162	0,0525	-0,8160	0,00	-0,02
C	0,6302	-0,7787	0,6300	-0,7785	-0,03	-0,03
D	0,1425	-1,4699	0,1425	-1,4696	0,00	-0,02
E	0,5776	-1,4474	0,5775	-1,4471	-0,02	-0,02
F	0,2551	-1,8862	0,2550	-1,8857	-0,04	-0,03
G	0,4876	-1,8787	0,4875	-1,8782	-0,02	-0,03
H	0,3751	-2,0274	0,3750	-2,0269	-0,03	-0,02
I	0,3751	-2,0274	0,3750	-2,0269	-0,03	-0,02
J	0,4951	-1,8862	0,4950	-1,8857	-0,02	-0,03
K	0,2626	-1,8787	0,2625	-1,8782	-0,04	-0,03
L	0,6077	-1,4699	0,6075	-1,4696	-0,03	-0,02
M	0,1725	-1,4474	0,1725	-1,4471	0,00	-0,02
N	0,6977	-0,8162	0,6975	-0,8160	-0,03	-0,02
O	0,1200	-0,7787	0,1200	-0,7785	0,00	-0,03
P	0,7502	0,0000	0,7500	0,0000	-0,03	-

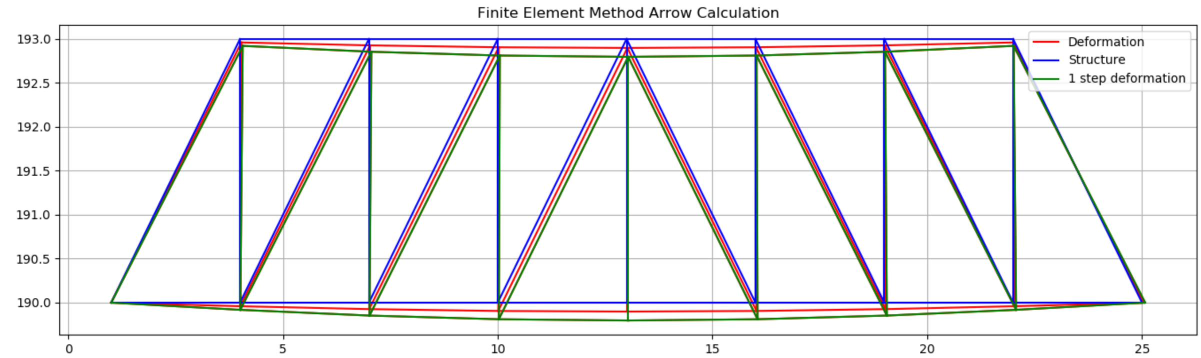
With the results obtained between the arrows obtained in each software, it is possible to verify that SATE presents results with a small percentage difference in relation to SAP2000 v.15, when the linear analysis of the structure is performed. This difference may be due to some disparity in the application of the Finite Element Method. However, the difference is too small to be considered relevant.

5.2 Efficiency of nonlinear SATE analysis

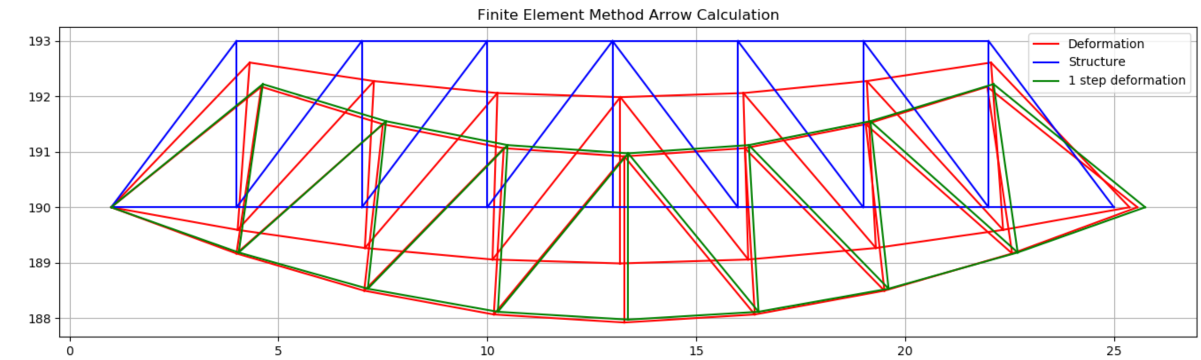
The nonlinear analysis of the structural behavior was obtained by applying the loads (3.00 kN, 10.00 kN and 100.00 kN) divided into 2 and 100 portions, in each case. This configuration simulates the progressive application of the load to the structure. SATE can graphically express the displacement progression for each load parcel. For a division into two portions of the loads applied, Fig. 7 shows the displacement of the structure.



(a)



(b)



(c)

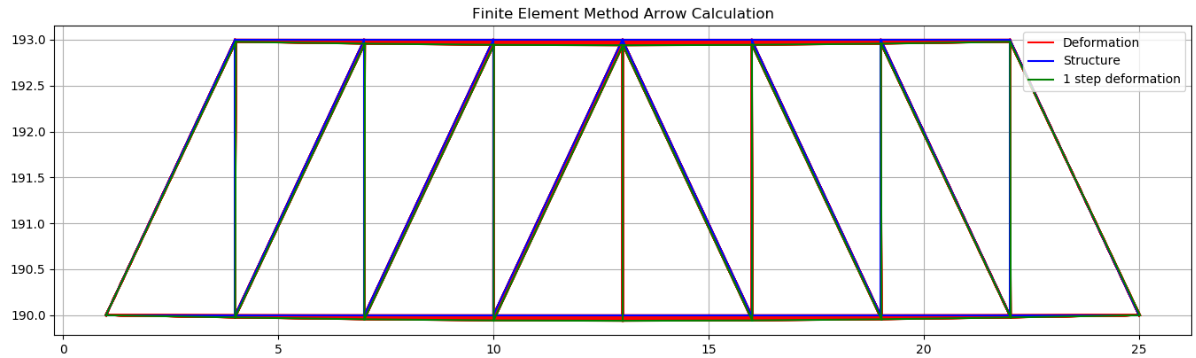
Figure 7. Deformed from the structure by SATE with two load increments for P = 3.00 kN (a), P = 10.00 kN (b) and P = 100.00 kN (c)

The results obtained by SATE after completion of the structure processing, understood as the final displacement values, can be seen in Table 4.

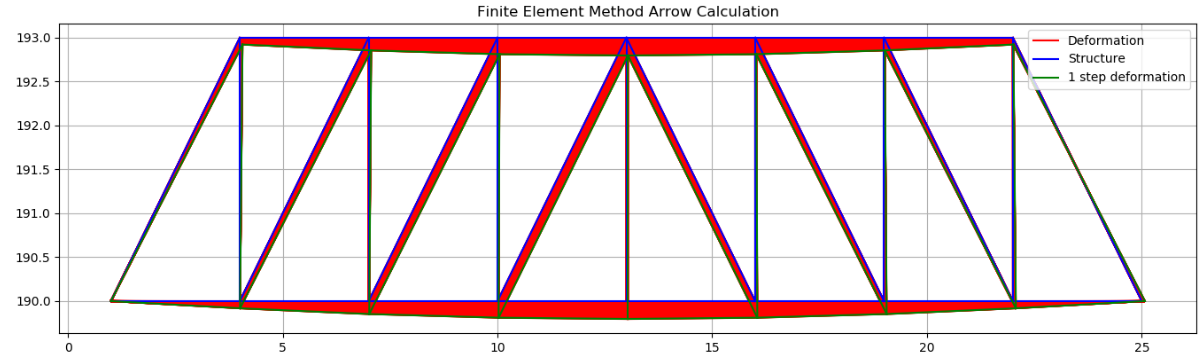
Table 4. SATE arrow values for two load increments

Node	Arrows obtained (m)					
	P=3.00 kN		**P=10.00 kN**		**P=100.00 kN**	
	Direction X	Direction Y	Direction X	Direction Y	Direction X	Direction Y
A	0,0000	0,0000	0,0000	0,0000	0,0000	0,0000
B	0,0015	-0,0245	0,0047	-0,0818	-0,0011	-0,8338
C	0,0189	-0,0234	0,0626	-0,0783	0,5944	-0,8259
D	0,0042	-0,0441	0,0134	-0,1473	0,0565	-1,5053
E	0,0173	-0,0435	0,0570	-0,1452	0,5029	-1,5008
F	0,0076	-0,0566	0,0246	-0,1890	0,1589	-1,9343
G	0,0145	-0,0564	0,0479	-0,1883	0,3965	-1,9340
H	0,0112	-0,0608	0,0366	-0,2032	0,2821	-2,0793
I	0,0112	-0,0609	0,0366	-0,2032	0,2821	-2,0821
J	0,0148	-0,0566	0,0486	-0,1890	0,4053	-1,9343
K	0,0078	-0,0564	0,0253	-0,1883	0,1677	-1,9340
L	0,0181	-0,0441	0,0598	-0,1473	0,5078	-1,5053
M	0,0051	-0,0435	0,0162	-0,1452	0,0614	-1,5008
N	0,0208	-0,0245	0,0685	-0,0818	0,5654	-0,8338
O	0,0035	-0,0234	0,0105	-0,0783	-0,0301	-0,8259
P	0,0223	0,0000	0,0732	0,0000	0,5642	0,0000

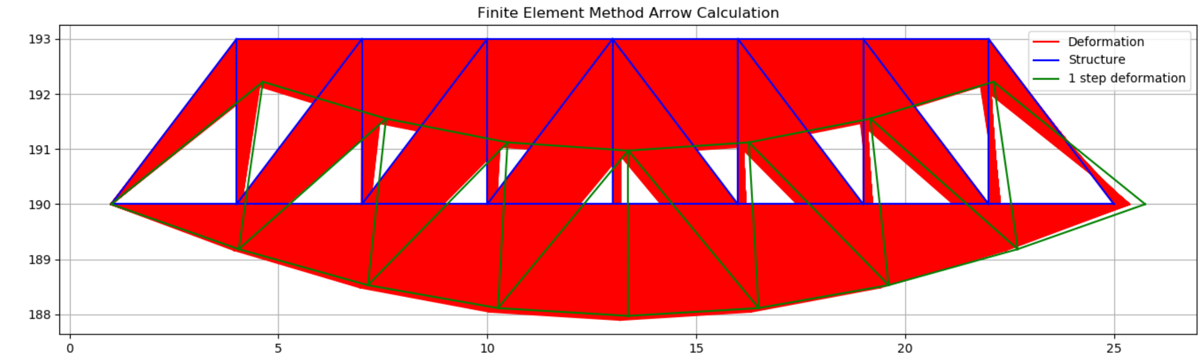
In order to analyze the behavior of the displacements with the increase of the load division for the loads of 3.00 kN, 10.00 kN and 100.00 kN were partitioned in 100 portions. The behavior of the truss obtained via SATE is shown in the following figure.



(a)



(b)



(c)

Figure 8. Deformed from the structure by SATE with two load increments for P = 3.00 kN (a), P = 10.00 kN (b) and P = 100.00 kN (c)

For 100 load increments, the results obtained by SATE after completion of frame processing can be seen in Table 5.

Table 5. SATE arrow values for 100 load increments

Node	**Arrows obtained (m)**					
	P=3.00 kN		**P=10.00 kN**		**P=100.00 kN**	
	Direction X	Direção Y	Direction X	Direção Y	Direction X	Direção Y
A	0,0000	0,0000	0,0000	0,0000	0,0000	0,0000
B	0,0015	-0,0245	0,0042	-0,0819	-0,0565	-0,8346
C	0,0188	-0,0234	0,0623	-0,0787	0,5481	-0,8585
D	0,0041	-0,0441	0,0126	-0,1476	-0,0341	-1,5143
E	0,0172	-0,0435	0,0563	-0,1457	0,4190	-1,5290
F	0,0075	-0,0566	0,0236	-0,1894	0,0564	-1,9526
G	0,0145	-0,0564	0,0470	-0,1888	0,2976	-1,9599
H	0,0111	-0,0609	0,0357	-0,2036	0,1821	-2,1006
I	0,0111	-0,0609	0,0357	-0,2036	0,1821	-2,1065
J	0,0147	-0,0566	0,0477	-0,1894	0,3079	-1,9526
K	0,0077	-0,0564	0,0244	-0,1888	0,0666	-1,9599
L	0,0180	-0,0441	0,0588	-0,1476	0,3984	-1,5143
M	0,0050	-0,0435	0,0151	-0,1457	-0,0548	-1,5290
N	0,0207	-0,0245	0,0672	-0,0819	0,4208	-0,8346
O	0,0033	-0,0234	0,0091	-0,0787	-0,1838	-0,8585
P	0,0222	0,0000	0,0714	0,0000	0,3643	0,0000

SAP2000 v.15 has two types of geometric nonlinearity analysis, the “P-Delta” and the “P-Delta plus Large Displacements", this second is used when there are large displacements, such as the arrow resulting from the 100kN load. Thus, the "P-Delta" analysis was applied for the 3 and 10 kN loads and the "P-Delta plus Large Displacements" analysis for the 100.00 kN load was applied. This obtained the results presented in Table 6.

Table 6. Arrow Values Considering Nonlinearity by SAP2000 v.15

Node	Arrows obtained (m)					
	P=3.00 kN		**P=10.00 kN**		**P=100.00 kN**	
	Direction X	Direção Y	Direction X	Direção Y	Direction X	Direção Y
A	0,0000	0,0000	0,0000	0,0000	0,0000	0,0000
B	0,0016	-0,0245	0,0054	-0,0821	-0,0571	-0,8448
C	0,0190	-0,0234	0,0636	-0,0784	0,5596	-0,8750
D	0,0043	-0,0442	0,0144	-0,1478	-0,0364	-1,5318
E	0,0174	-0,0435	0,0583	-0,1456	0,4265	-1,5526
F	0,0077	-0,0567	0,0258	-0,1896	0,0526	-1,9716
G	0,0147	-0,0565	0,0492	-0,1889	0,2998	-1,9855
H	0,0113	-0,0609	0,0379	-0,2037	0,1770	-2,1132
I	0,0113	-0,0609	0,0379	-0,2038	0,1770	-2,1247
J	0,0149	-0,0567	0,0500	-0,1896	0,3014	-1,9716
K	0,0079	-0,0565	0,0265	-0,1889	0,0543	-1,9855
L	0,0183	-0,0442	0,0613	-0,1478	0,3904	-1,5318
M	0,0052	-0,0435	0,0175	-0,1456	-0,0724	-1,5526
N	0,0210	-0,0245	0,0704	-0,0821	0,4111	-0,8448
O	0,0036	-0,0234	0,0122	-0,0784	-0,2056	-0,8750
P	0,0226	0,0000	0,0758	0,0000	0,3540	0,0000

To verify the effectiveness of this simplified geometric nonlinearity approach made by SATE, the values obtained by this code, in two and one hundred load divisions, were compared to the results obtained in SAP2000 v.15, where this comparison is shown in the Tables 7 and 8.

Table 7. SATE efficiency over SAP2000 v.15 for two load increments

Node	Difference between arrows obtained in SATE and SAP2000 v.15 (%)					
	P=3.00 kN		**P=10.00 kN**		**P=100.00 kN**	
	Direction X	Direction Y	Direction X	Direction Y	Direction X	Direction Y
A	-	-	-	-	-	-
B	-6,25	0,00	-12,96	-0,37	-98,07	-1,30
C	-0,53	0,00	-1,57	-0,13	6,22	-5,61
D	-2,33	-0,23	-6,94	-0,34	-255,22	-1,73
E	-0,57	0,00	-2,23	-0,27	17,91	-3,34
F	-1,30	-0,18	-4,65	-0,32	202,09	-1,89
G	-1,36	-0,18	-2,64	-0,32	32,25	-2,59
H	-0,88	-0,16	-3,43	-0,25	59,38	-1,60
I	-0,88	0,00	-3,43	-0,29	59,38	-2,00
J	-0,67	-0,18	-2,80	-0,32	34,47	-1,89
K	-1,27	-0,18	-4,53	-0,32	208,84	-2,59
L	-1,09	-0,23	-2,45	-0,34	30,07	-1,73
M	-1,92	0,00	-7,43	-0,27	-184,81	-3,34
N	-0,95	0,00	-2,70	-0,37	37,53	-1,30
O	-2,78	0,00	-13,93	-0,13	-85,36	-5,61
P	0,3540	0,0000	-3,43	-	59,38	-

Table 8. SATE efficiency over SAP2000 v.15 for one hundred load increments

Node	Difference between arrows obtained in SATE and SAP2000 v.15 (%)					
	P=3.00 kN		P=10.00 kN		P=100.00 kN	
	Direction X	Direção Y	Direction X	Direção Y	Direction X	Direção Y
A	-	-	-	-	-	-
B	-6,25	0,00	-22,22	-0,24	-1,05	-1,21
C	-1,05	0,00	-2,04	0,38	-2,06	-1,89
D	-4,65	-0,23	-12,50	-0,14	-6,32	-1,14
E	-1,15	0,00	-3,43	0,07	-1,76	-1,52
F	-2,60	-0,18	-8,53	-0,11	7,22	-0,96
G	-1,36	-0,18	-4,47	-0,05	-0,73	-1,29
H	-1,77	0,00	-5,80	-0,05	2,88	-0,60
I	-1,77	0,00	-5,80	-0,10	2,88	-0,86
J	-1,34	-0,18	-4,60	-0,11	2,16	-0,96
K	-2,53	-0,18	-7,92	-0,05	22,65	-1,29
L	-1,64	-0,23	-4,08	-0,14	2,05	-1,14
M	-3,85	0,00	-13,71	0,07	-24,31	-1,52
N	-1,43	0,00	-4,55	-0,24	2,36	-1,21
O	-8,33	0,00	-25,41	0,38	-10,60	-1,89
P	-1,77	-	-5,80	-	2,91	-

The negative values of the SATE efficiency table express that the SATE arrows values were lower than SAP2000 v.15. After viewing these tables some observations can be made, one of them being the fact that the percentage difference is smaller for smaller loads, and we can also see that the greater the number of steps of load application, the smaller the percentage difference, especially in Y axis. In general, the X axis is the one with the largest percentage difference. This is also due to the fact that this axis has the smallest displacements in absolute values, causing even small displacements to generate a large percentage difference.

5.3 Difference between linear and nonlinear arrows

Now for the third comparative analysis one must evaluate the percentage difference between the linearly calculated and the nonlinearly calculated arrows. This analysis was done first for SATE.

Table 9. Comparison between SATE linear and nonlinear arrows for two load increments

Nó	Percentage diferences between arrows (%)					
	P=3.00 kN		P=10.00 kN		P=100.00 kN	
	X	Y	X	Y	X	Y
A	-	-	-	-	-	-
B	-6,25	0,00	-11,32	0,25	-102,10	2,18
C	0,00	0,00	-0,63	0,64	-5,65	6,09
D	-2,33	0,00	-5,63	0,20	-60,35	2,43
E	0,00	0,23	-1,38	0,35	-12,92	3,71
F	0,00	0,00	-3,53	0,21	-37,69	2,58
G	-0,68	0,18	-1,84	0,27	-18,67	2,97
H	-0,88	0,00	-2,40	0,25	-24,77	2,59
I	-0,88	0,16	-2,40	0,25	-24,77	2,72
J	0,00	0,00	-1,82	0,21	-18,12	2,58
K	-1,27	0,18	-3,80	0,27	-36,11	2,97
L	-0,55	0,00	-1,48	0,20	-16,41	2,43
M	-1,92	0,23	-6,36	0,35	-64,41	3,71
N	-0,48	0,00	-1,72	0,25	-18,94	2,18
O	-2,78	0,00	-12,50	0,64	-125,08	6,09
P	-0,89	-	-2,40	-	-24,77	-

Table 10. Comparison between SATE linear and nonlinear arrows for one hundred load increments

Nó	**Percentage diferences between arrows (%)**					
	P=3.00 kN		**P=10.00 kN**		**P=100.00 kN**	
	X	Y	X	Y	X	Y
A	-	-	-	-	-	-
B	-6,25	0,00	-20,75	0,37	-207,62	2,28
C	-0,53	0,00	-1,11	1,16	-13,00	10,28
D	-4,65	0,00	-11,27	0,41	-123,93	3,04
E	-0,58	0,23	-2,60	0,69	-27,45	5,66
F	-1,32	0,00	-7,45	0,42	-77,88	3,55
G	-0,68	0,18	-3,69	0,53	-38,95	4,35
H	-1,77	0,16	-4,80	0,44	-51,44	3,64
I	-1,77	0,16	-4,80	0,44	-51,44	3,93
J	-0,68	0,00	-3,64	0,42	-37,80	3,55
K	-2,53	0,18	-7,22	0,53	-74,63	4,35
L	-1,10	0,00	-3,13	0,41	-34,42	3,04
M	-3,85	0,23	-12,72	0,69	-131,77	5,66
N	-0,96	0,00	-3,59	0,37	-39,67	2,28
O	-8,33	0,00	-24,17	1,16	-253,17	10,28
P	-1,33	-	-4,80	-	-51,43	-

Finally this same analysis was done with the results of the SAP2000 v.15 arrows. Table 11 shows the percentage difference between the arrow values obtained with linear and nonlinear geometric analysis.

Table 11. Comparison between SAP2000 linear and nonlinear arrows

Nó	**Percentage diferences between arrows (%)**					
	P=3.00 kN		**P=10.00 kN**		**P=100.00 kN**	
	X	Y	X	Y	X	Y
A	-	-	-	-	-	-
B	0,00	0,00	1,89	0,61	-208,76	3,50
C	0,53	0,00	0,95	0,64	-11,20	12,37
D	0,00	0,23	0,70	0,54	-125,54	4,21
E	0,58	0,23	0,87	0,62	-26,16	7,27
F	0,00	0,18	1,18	0,53	-79,38	4,53
G	0,68	0,18	0,82	0,53	-38,52	5,68
H	0,00	0,16	1,07	0,49	-52,81	4,23
I	0,00	0,16	1,07	0,54	-52,81	4,80
J	0,00	0,18	1,01	0,53	-39,12	4,53
K	0,00	0,18	0,76	0,53	-79,32	5,68
L	0,55	0,23	0,82	0,54	-35,76	4,21
M	0,00	0,23	1,16	0,62	-141,97	7,27
N	0,48	0,00	2,18	0,61	-41,08	3,50
O	0,00	0,00	1,67	0,64	-271,33	12,37
P	0,44	-	1,07	-	-52,81	-

It is noticed that the X coordinate is very variable, especially for 100.00 kN loading, this is because when the load is applied at once (linear analysis) most of the nodes are displaced in the positive direction of the X axis, even when account is taken of geometric nonlinearity, this abrupt deformation does not occur, on the contrary, the structure has longer time to deform evenly, causing some nodes to be displaced in the negative direction of the X axis due to the more accommodating inclination of the structure caused by rotation, this phenomenon can be easily seen by looking at the nodes of any of the graphs with geometric nonlinearity presented. In addition, excessive Y-axis deformation in the case of 100.00 kN load "pulls" all points a little further to the negative direction of the X axis.

Analyzing the tables it can also be noted that the percentage difference of arrow on the Y axis, is bigger the greater the number of steps applied, it also increases with the increase of the applied load, a fact that can be observed comparing the results for the loads of 3.00, 10.00 and 100.00 kN.

6 CONCLUSION

After exposition of all results it can be noted that the geometric nonlinearity did not have such a relevant impact for the loading of 3.00 kN, where it was observed that the largest percentage difference of the Y axis was 0.23%, this happens mainly. Due to the fact that the load and displacement are low, in this case nonlinearity can be scorned.

We also saw that the largest percentage difference on the Y axis between the nonlinear analysis of SATE and SAP2000 v.15 was for the load of 100.00 kN of 1.89%. It was expected that there would be a difference between the SATE and SAP2000 v.15 values, since SAP2000 v.15 uses continuous methods while SATE uses a more simplified approach, just dividing the load and applying after each deformation of the structure. However, perhaps this percentage difference value could be reduced by increasing the number of divisions in which the load would be applied.

For large displacements the analysis of the structure taking into account the geometric nonlinearity becomes practically mandatory, as we can see in the nonlinear analysis with load of 100.00 kN, where the percentage difference between the linear and nonlinear methods reaches 10.28% in SATE and 12.37% in SAP2000 v.15.

In short, the concept of geometric nonlinear analysis is well applied when using the “step analysis” which has been explained and exemplified here. This kind of step analysis can be very well implemented if we take into account the loading phases that happen in a work, where there are the own weight, construction, design and so on. It is known that for a structure to deform it needs time, so in a real situation, it would be of no use to apply the load per step, but with very short time interval between the applications of each step, so use the steps of projected loading may further optimize the step analysis proposed in this work.

References

[1] LACERDA, E. G. M. Nonlinear lattice analysis by positional finite element method. 2014. 107 f. Dissertation (Master) - Technology Center, Federal University of Rio Grande do Norte, Natal, 2014.
[2] BRAZILIAN ASSOCIATION OF TECHNICAL STANDARDS. NBR 8800: Design of steel structures and mixed steel and concrete structures of buildings. Rio de Janeiro, 2008.
[3] CUNHA, P. F. L. Computational routines for nonlinear geometric analysis of lattice structures. 2017. 156f. Dissertation (Master) - Center for Technology and Geosciences, Federal University of Pernambuco, Recife, 2017.
[4] BENJAMIN, A. C. Geometric nonlinear analysis of three-dimensional frames by the finite element method.1982. 229 f. Dissertation (Master) - COPPE, Federal University of Rio de Janeiro, Rio de Janeiro, 1982.
[5] PINTO, R. S.; RAMALHO, M. A. Physical and geometric nonlinearity in the design of usual reinforced concrete buildings. São Carlos: EESC-USP Publication, 2002.
[6] EMPTY, Luiz Eloy. Finite element method in structural analysis. Rio de Janeiro: Elsevier, 2011.