

# A FREE COMPUTER PROGRAM FOR THE ANALYSIS OF 3-DIMENSIONAL FRAMES.

**VICTOR MATEI PETRESCU** - Teaching assistant, PhD, Technical University of Civil Engineering Bucharest, Faculty of Hydrotechnics, e-mail: matei@ppm.ro

**Abstract:** A free and open source computer program which calculates displacements, reactions and sectional forces for 3-dimensional frames was developed. Its source code and documentation can be downloaded from <http://bars3d.sourceforge.net>. The program was developed mainly with the purpose of obtaining features not present in other such programs available today, like: ease of compilation on as many operating systems as possible, the declaration of the input data in the form of a text file with a simple and easy to read structure, clear and useful error messages when the structure of the input file is incorrect, a graphical interface for displaying diagrams which can also show the deformed or undeformed structure from any angle and perspective, the possibility to declare elements with arbitrarily variable sections and distributed loads without explicitly introducing supplementary nodes and also other features, which can not be mentioned in the abstract, due to the limited space. These features, the way in which they were achieved and the results of some analysis examples are presented in the article.

**Rezumat:** A fost realizat un program de calcul liber și cu sursă deschisă pentru determinarea deplasărilor, reacțiunilor și eforturilor secționale ale unor sisteme de bare tridimensionale. Codul sursă și documentația pot fi descărcate de la adresa <http://bars3d.sourceforge.net>. Programul a fost făcut în special cu scopul de a se obține facilități care nu sunt oferite de alte astfel de programe disponibile în prezent, precum: compilare fără dificultăți pe cât mai multe sisteme de operare, declararea datelor de intrare sub forma unui fișier text cu o structură simplă și ușor de citit, mesaje de eroare clare și utile în cazul în care structura fișierului cu date de intrare nu e corectă, o interfață grafică pentru afișarea diagramelor care să permită vizualizarea formei deformate sau nedeformate a structurii din orice unghi și perspectivă, posibilitatea de a declara elemente cu secțiuni și încărcări distribuite având orice fel de variație fără introducerea unor noduri suplimentare precum și alte facilități pentru menționarea cărora spațiul disponibil în rezumat nu este suficient. În articol sunt prezentate caracteristicile programului, modul în care s-au obținut și rezultatele unor exemple de calcul.

**Keywords:** structure, civil, engineering, frame, free

## 1. INTRODUCTION

At present (2013), a large number of computer programs based on the finite element method have been developed for the analysis of frames and other structures encountered in civil engineering [1,2]. Although these programs were made for approximately the same purpose, very significant differences exist between them, concerning their capabilities as well as their user interfaces. Choosing the appropriate program for a certain problem can therefore be very difficult. Considering the disadvantages of the already available programs (such as restrictive licenses and/or difficulties in their utilization), a new program was developed, which can be downloaded from <http://bars3d.sourceforge.net> under the name of "bars3d-2.0.0" and is available under the GNU GPL license ( <http://www.gnu.org> ). Its main features are presented and exemplified below.

## 2. OVERVIEW OF THE MAIN FEATURES

The program computes displacements, sectional forces and reactions for 3-dimensional frames, such as those illustrated in Figure 1. As can be deduced from Figure 1, the deformed or undeformed structures can be viewed from any angle and perspective. A library called "SDL", available at <http://www.libsdl.org>, was used for the graphical representations. This library has versions for a large number of operating systems and was the only library used, besides the standard C library. Therefore, the program can be compiled almost without modifications on any operating system for which a version of "SDL" exists (a partial list of these operating systems can be found at <http://www.libsdl.org>).

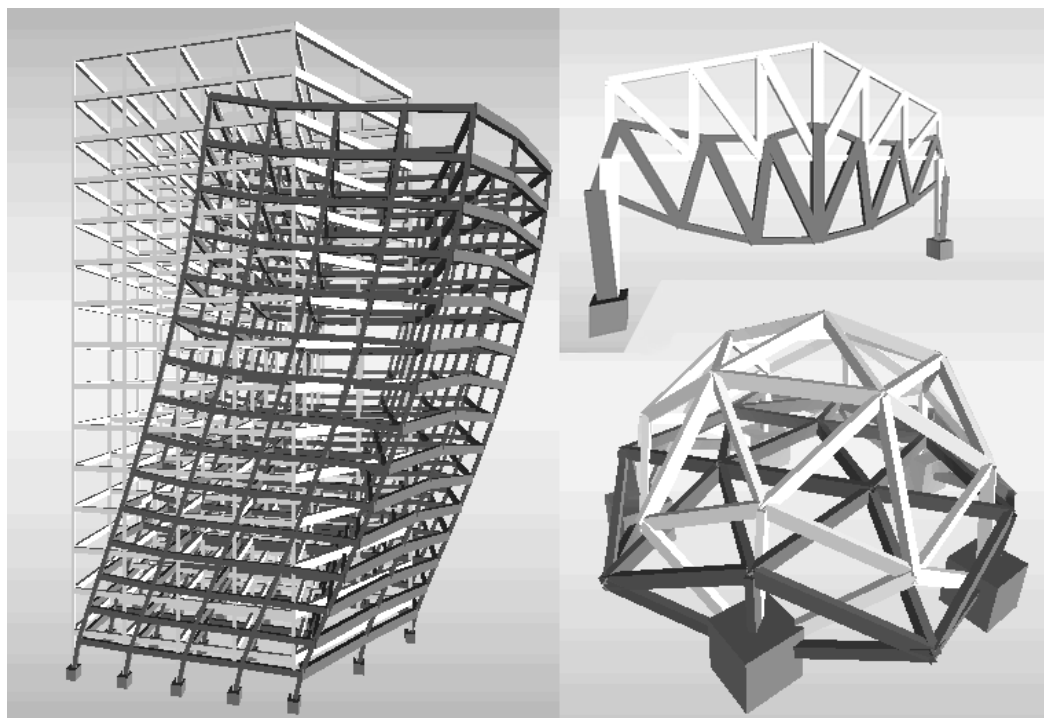


Figure 1 - 3D representations of the undeformed and deformed shapes of several structures analysed with bars3d-2.0.0

The finite elements used are of the type shown in Figure 2 - linear elements, with 2 nodes and 6 degrees of freedom at each node. An element can be composed of several smaller elements and therefore have several intermediary nodes (as shown in Figure 5). A stiffness matrix corresponding to the degrees of freedom of all intermediary nodes is calculated for each element, then a global stiffness matrix in which only the nodes at the ends of the elements are accounted for is calculated from the stiffness matrices of the elements and after determining the displacements of the end nodes, the stiffness matrices of the elements are used again for the calculation of displacements and forces at the intermediary nodes. This method allows each element to have arbitrarily variable section parameters and distributed load, while avoiding an excessively large global stiffness matrix.

The parameters of the structure to be analysed must be written into a text file. Details concerning the declaration of these parameters can be found in the program's documentation. Because one of the aims was to obtain a program as easy to use as possible, some limitations exist, such as:

- Only one condition can be specified for all 6 degrees of freedom of a node; they can either be free or blocked. This is compensated by the fact that each element can be divided into an arbitrarily large number of smaller elements, so hinges can be modelled with good approximation if necessary, as will be exemplified below.
- The distributed load on each element is considered to act only in the direction of and opposed to global axis X. This limitation can be easily eliminated with a few modifications in the source code of the program (which, as specified in the introduction, is free and open source). A modelled structure must therefore always be positioned relative to the global axes as shown in Figure 3.

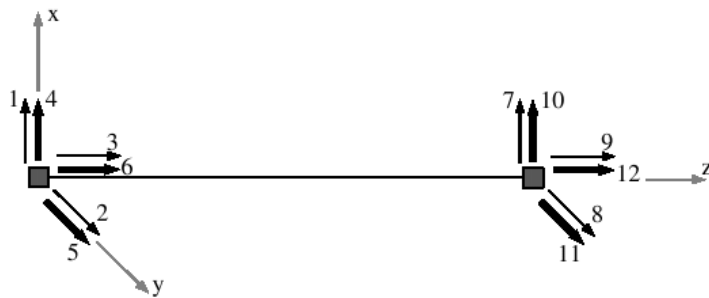


Figure 2 - Finite element used

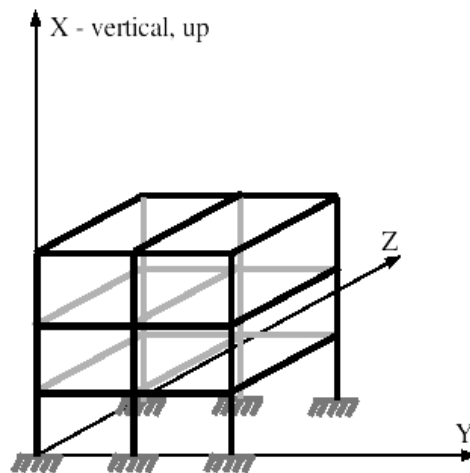


Figure 3 - Position of the structure, relative to the global axes

Before the analysis of the structure is started, the input file is checked for a number of possible errors and error messages are displayed, which show the problem encountered its location. A few such error messages are exemplified below:

*Error: 'dataframe' line 41 - section '10' doesn't exist*

*Error: 'dataframe' line 40 - integer expected, got '21.5'*

*Error: 'dataframe' line 27 - sections should have been declared before*

Although very simple and useful, such a feature was not found in any other structural analysis programs tried by the author of this article.

### 3. ANALYSIS EXAMPLES

#### 3.1. Simply supported beam with uniformly distributed load

An analysis was performed for the beam shown in Figure 4. Because hinges and simple supports can not be modelled directly with the program presented here, they were approximated by reducing the stiffnesses of the segments at the ends of the beam. The model which was actually analysed is shown in Figure 5. Generally, in such a case, good results were found to be obtained by dividing the element into  $n$  segments and using  $n^2$  times smaller inertial moments for segments 1 and  $n$ .

The following parameters were considered:

- distributed load:  $q = 10 \text{ kN/m}$ , acting in the direction of and opposed to local axis  $x$ ;
- length:  $l = 5 \text{ m}$ ;
- sectional area:  $A = 0.125 \text{ m}^2$ ;
- shear areas:  $A_x = A_y = 0.8 \cdot A = 0.1 \text{ m}^2$ ;
- inertial moments:  $I_x = 0.000651 \text{ m}^4$ ,  $I_y = 0.002604 \text{ m}^4$ ;
- elasticity moduli:  $E = 21000000 \text{ kN/m}^2$ ,  $G = 8076923 \text{ kN/m}^2$ .

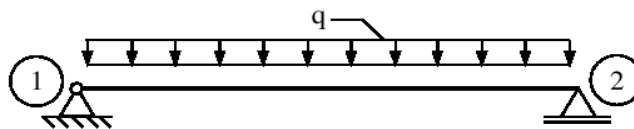


Figure 4 - Simply supported beam

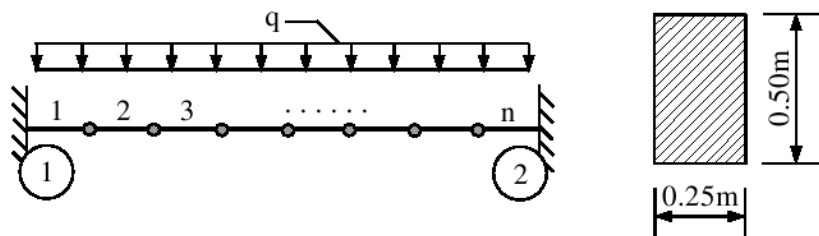


Figure 5 - Approximate model of the beam shown in Figure 4

The beam was divided into 100 segments and  $I_y$  was reduced  $100^2 = 10000$  times for segments 1 and  $n$ . The diagrams of bending moments and displacements obtained are shown in Figure 6. The contents of the input file is also shown below:

```
nodes 2
1 1 0 0 0
2 1 0 5 0
sections 2
sc1 1 21000000 0.3 0.125 0.00001 0.000651 0.002604 0.1 0.1
sc2 1 21000000 0.3 0.125 0.00001 0.000651 0.000002604 0.1 0.1
bars 1
b1 1 2 3 1 sc2 10 98 sc1 10 1 sc2 10
```

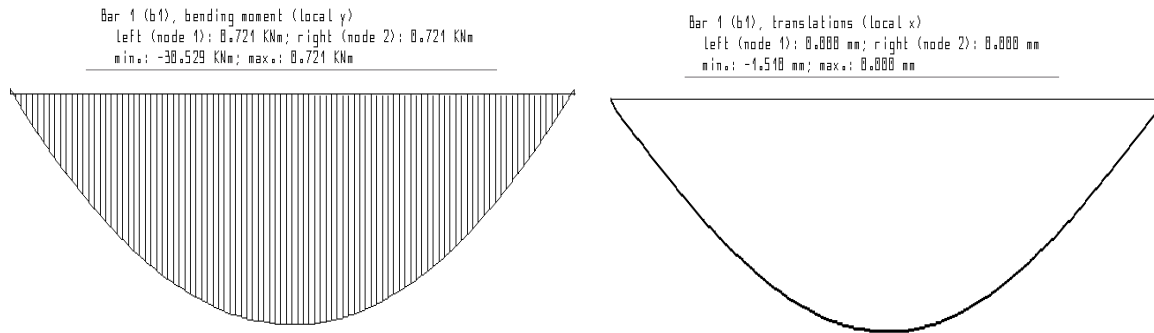
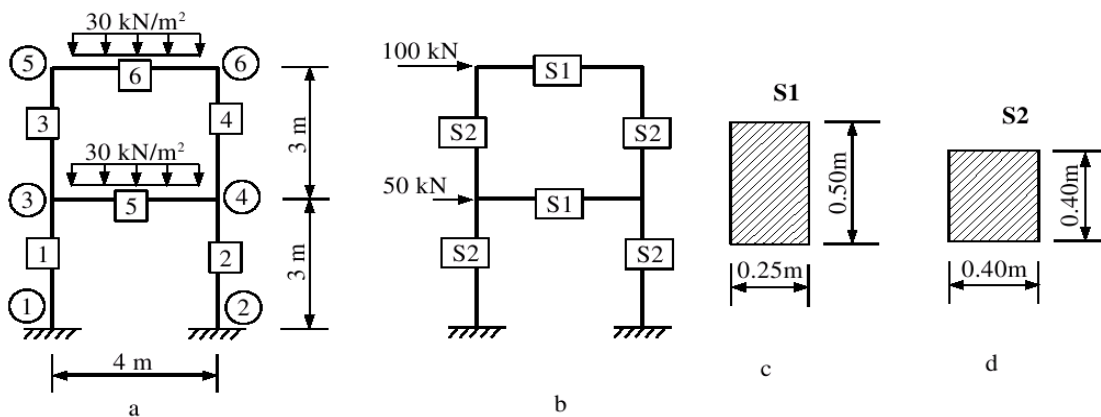


Figure 6 - Diagrams of bending moments and vertical displacements for the model in Figure 5

The maximum bending moment for the beam in Figure 4 is  $M_{\max} = ql^2/8 = 31.25 \text{ kNm}$ , which is 2.4% larger than the value obtained with the program for the approximate model in Figure 5, 30.529 kNm, and the maximum displacement for the beam in Figure 4 is  $u_{\max} = 5ql^4/(384EI_y) + ql^2/(8GA_x) = 1.527 \text{ mm}$ , 1.1% larger than the value obtained with the program and shown in Figure 6, 1.510 mm. Thus, hinges can be modelled with good approximation, especially considering that perfect hinges are seldom encountered in civil engineering structures.

### 3.2. Linear and nonlinear analyses of a frame

Two analyses, one linear and one nonlinear, were performed for the frame in Figure 7. In the nonlinear analysis, some plastic moments were considered for section S1 (S2 was unmodified).



a - nodes, elements and distributed loads; b - sections and concentrated loads; c and d - sections S1 and S2

Figure 7 - Frame for which some analyses were performed

The moment-curvature diagram for an element with nonlinear behaviour is shown in Figure 8. Each of the  $n$  elements which form a large element (as shown in Figure 5) can have its own plastic moments. If nonlinear analysis is selected from the program's interface, the loads are applied in 2 phases: the distributed loads in the first phase and the concentrated loads in the second phase. The loads in each phase are applied in a number of steps and the stiffnesses of the elements are modified at each step according to the diagram shown in Figure 8. Because plastic zones with finite lengths are modelled in this way, rather than plastic hinges, the stiffness of an element can not be reduced to 0.

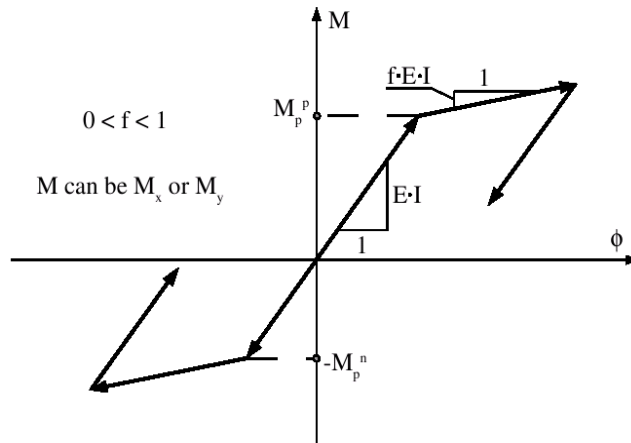


Figure 8 - Moment-curvature diagram for an element with nonlinear behaviour

The parameters used in the 2 analyses for sections S1 and S2 were:

- inertial moments:  $I_y^{S1} = 0.002604 \text{ m}^4$ ,  $I_y^{S2} = 0.00213 \text{ m}^4$ ;
- sectional areas:  $A^{S1} = 0.125 \text{ m}^2$ ,  $A^{S2} = 0.16 \text{ m}^2$ ;
- shear areas:  $A_x^{S1} = 0.1 \text{ m}^2$ ;  $A_x^{S2} = 0.128 \text{ m}^2$ ;
- elasticity moduli:  $E = 21000000 \text{ kN/m}^2$ ,  $G = 8076923 \text{ kN/m}^2$ ;
- plastic moments for S1:  $M_{py}^p = 70 \text{ kNm}$  (positive);  $M_{py}^n = 40 \text{ kNm}$  (negative);
- reduction factor for the inertial moment (as shown in Figure 8):  $f = 0.0001$ .

Diagrams of bending moments, translations and rotations along element 5 in Figure 7 are shown in Figure 9 for the linear analysis and in Figure 10 for the nonlinear analysis.

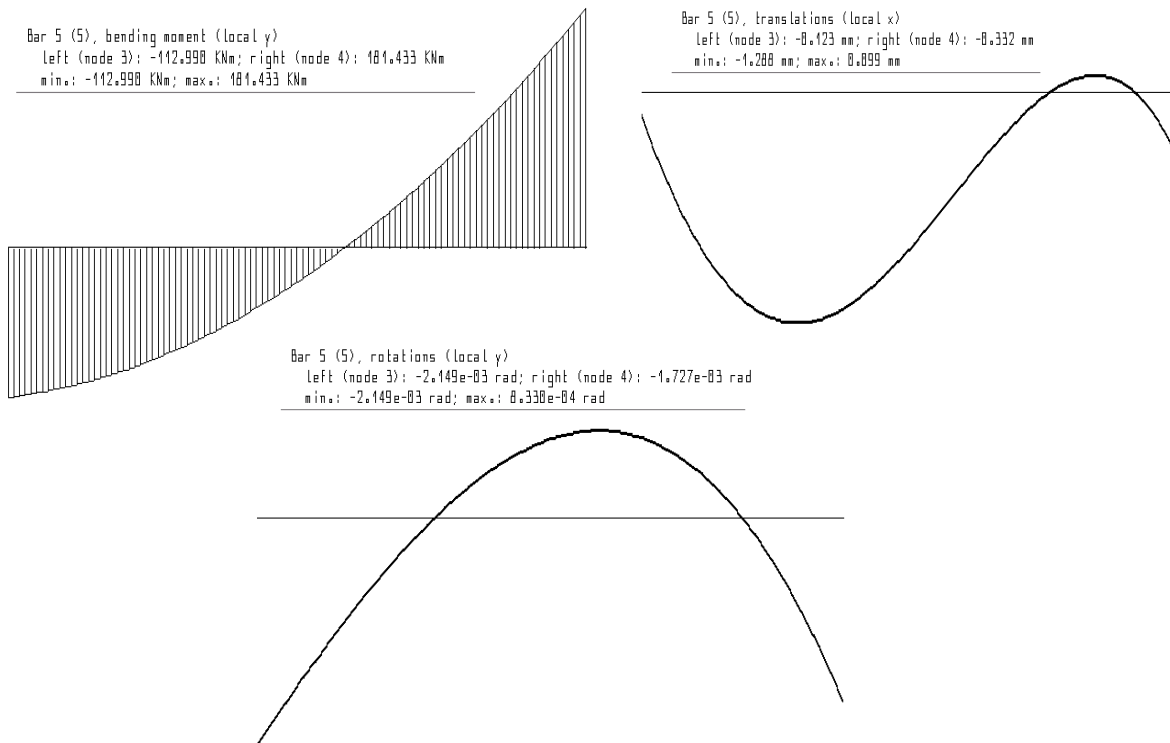


Figure 9 - Bending moments, translations and rotations for element 5 in Figure 7, from the linear analysis

In the nonlinear analysis the loads were applied in 2 phases, as described above. In each phase, the loads were applied in 100 steps. Figure 10 also shows the horizontal translation of node 5 during the 100 steps of the second phase.

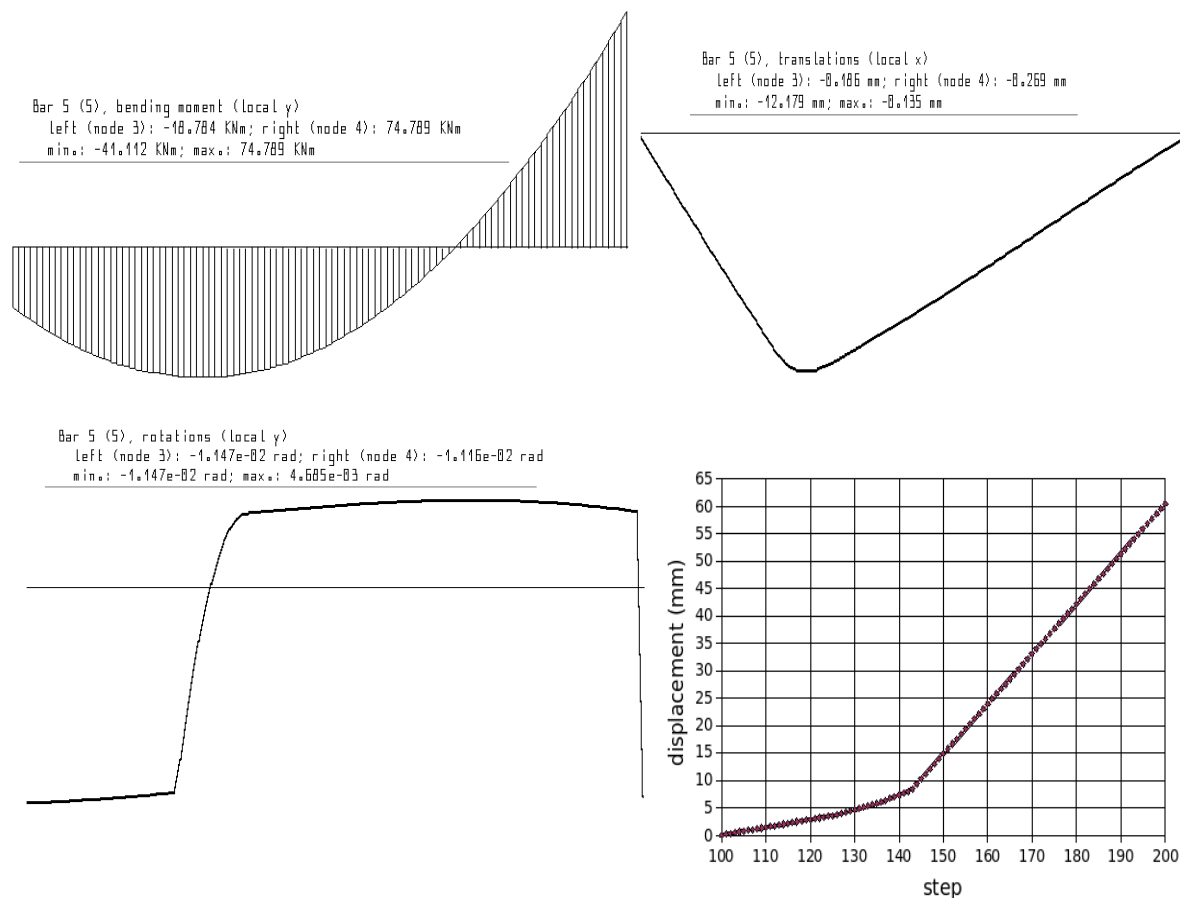


Figure 10 - Bending moments, translations and rotations for element 5 in Figure 7 and horizontal displacement of node 5 from Figure 7 during the second phase of the nonlinear analysis

## 4. CONCLUSIONS

The program presented in this article should be a viable alternative to other programs designed for static analyses of frames, considering its main advantages:

- It can perform both linear and nonlinear analyses;
- It was specifically designed to be easy to use, with a simple structure of the input file and an intuitive and practical interface;
- It is free and available under the GNU GPL license ( <http://www.gnu.org> ), so it can be used, studied and modified without any restriction and also distributed, modified or unmodified.

## REFERENCES

- [1] Wilson, E.L. - Three Dimensional Static and Dynamic Analysis of Structures - Computers and Structures Inc. Berkeley, California, USA, 2000
- [2] <http://frame3dd.sourceforge.net>