A matlab program for plastic analysis of 2-d frames

Marsel Nakuçi[1], Soen Sako[2]

[1]*Department of Civil Engineering, Faculty of civil engineering, Polytechnic University of Albania*
[2]*Department of Civil Engineering, Faculty of civil engineering, Polytechnic University of Albania*

ABSTRACT

This paper describes a program for plastic analysis of 2-d frames written in matlab programming language. The program is developed using the object oriented programming approach as an alternative to traditional procedural programming. A step by step incremental linear analysis is used for the solution. The two node frame element represents the finite element. The element stiffness matrix for frame element is constructed by superimposing both axial and bending stiffness. Hermite polynomials are used as interpolation functions for the solution of the finite element analysis. Element nodal forces and stiffness matrices are obtained using Gauss quadrature scheme as a numerical integration technique. This integration scheme for polynomial functions leads in a very accurate solution. Matlab library has served for the solution of systems of linear equations. The two most important classes in the program are the cstruct_lin_static_preprocess class which calculates all preprocess data and the cstruct_lin_plastic_postprocess class, which gets as input argument the object of preprocess class and calculates all postprocess data. A simple frame structure under lateral loads is presented as an example and solved by the program. The stress state represented by internal moment diagram, shear force diagram and axial force diagram is given for each step and the final step. The presentation of this data is in tabular form and graphical form. A plot of the deformed shape for every step is also provided by the program. The history of force-displacement and the plot of plastic hinges is displayed.

INTRODUCTION

Object oriented programming is the method used for developing the program. The classes are divided in two categories:

1. The preprocess classes
2. The postprocess classes

The classes that make the interface of the program are:

1. cnodes
2. cfr_ele
3. cstruct
4. cmaterial
5. crect_sec
6. cplastic_hinge
7. crestrain
8. cnodal_force
9. cfr_distributed_force

10. cfr_point_force
11. cstruct_lin_static_preprocess
12. cstruct_lin_plastic_postprocess

The objects of these classes will be initialized by the user.
Other classes part of the program are:

1. cfrsub_ele
2. cfrsub_lin_static_analytic
3. cfr_lin_static_analytic
4. cstruct_lin_static_analytic
5. cfrsub_lin_static_preprocess
6. cfr_lin_static_preprocess
7. cstruct_lin_static_preprocess
8. cfrsub_lin_static_postprocess
9. cfr_lin_static_postprocess
10. cstruct_lin_static_postprocess

Objects of these classes are constructed by the program. They are contained as properties with private SetAccess attribute, in the interface objects. This is also called inheritance by composition.

PROGRAM IMPLEMENTATION

Matlab software is needed to run the program. All the code written is contained in matlab ".m" class files.

Geometry, material, section, data

The geometry, material and section data is contained in **cmaterial**, **crect_sec**, **cplastic_hinge**, **cnodes**, **cfr_ele**, **cstruct** objects. These objects contain all the properties of the geometry like section dimensions, lengths of elements, angels of elements etc. The main properties of **cmaterial** are *masspervol*, *elasticitymod*, *poissonmod*. *Weightpervol*,*shearmod* are properties with 'Dependent' attribute. This means that when a reference occur in these properties a method will calculate and return their values. Having 'dependent attribute' also means that their values are not stored in object. The properties of **crect_sec** class are *width*,*height*,*material*. Dependent properties are *A*,*Iy*,*Iz*,*Ix*,,*iy*,*iz*. Propertie of **cplastic_hinge** is *fy* (contains the yielding stress). Node objects are contained in an array of **cnodes** objects. '*Coord*' is the main propertie of this class. It has the coordinate of the node. The main properties of **cfr_ele** class which constructs cfr_ele objects are *nodes*, *rect_sec*, *plastic_hinge*, *realease*, *mesh*. Dependent properties calculated by methods of object are *xcoord*, *ycoord*, *angle*, *length*. *Frsub_ele* is a SetAccess propertie assigned only by methods of the class. The **cstruct** class has two properties *fr_ele*, *nodes* which contain objects of cfr_ele and cnodes.

Boundary condition data

The essential boundary condition data (restrains) is located in **crestrain** object. The natural boundary condition data (forces) is contained inside **cnodal_force**, **cfr_point_force**, **cfr_distributed_force**. Properties of **crestrain** are *restrain* (3 by 1 vector, accepts 0 or 1. 0 for constrained degree of freedom 1 for free degree of freedom), *nodes* ,*fr_ele*. Properties of **cnodal_force** class are *force* (a 3 by 1 vector that contains the force) ,*nodes* (contains objects of cnodes class, it shows where the force is applied). Properties of **cfr_point_force** are *force* (3 by 1 vector) , *location* (accepts values from 0 to 1, location in relative distance), *elements* (objects of cfr_ele class). Properties of **cfr_distributed_force** are *force* (3 by 1

vector) , *location* (a 2 by 1 vector ,accepts values from 0 to 1, location in relative distance), *elements* (objects of cfr_ele class).

Preprocess data

All preprocess information like stiffness matrices, force vector etc. is contained inside object of **cstruct_lin_static_preprocess** class .

Postprocess data

All postprocess information like nodal displacement, internal forces etc. is contained inside object of **cstruct_lin_static_postprocess** class. The most important method of this class is 'calc(obj)'. This method makes all the calculation and stores the postprocess data. The flow chart of this method is presented below in Figure 1.

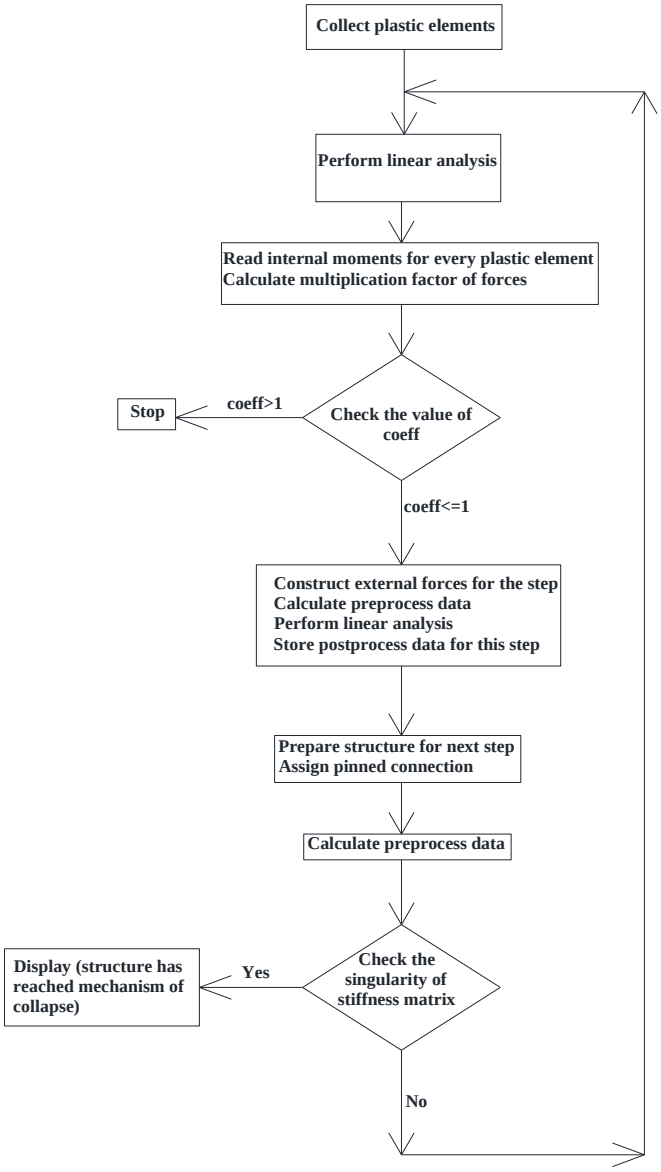

Figure 1 The flow chart of 'calc(obj)' method

The value of coeff is calculated as shown in Eq.(1)

$$\text{coeff} = \text{abs}\ \frac{\text{mpl}-\text{abs(M)}}{\text{m}} \tag{1}$$

where mpl is the plastic moment of the section, plastic hinges are of type 'M3' implying that axial stress is neglected. M is the internal moment from previous steps, m is the internal moment from current step, coeff is the multiplication factor of external forces.

EXAMPLE PROBLEM

A four story frame under lateral loads is considered. The dimensions of columns sections are 0.45x0.45m and dimensions of beam sections are 0.3x0.4m. The yielding stress is 4500kn/m^2 from Eq. (2). The story height is 3m and space between columns is 5 m. All elements releases are considered as fixed-fixed type and the structure is supported with fixed restrains. The node numbering and element numbering is shown in Figure 2

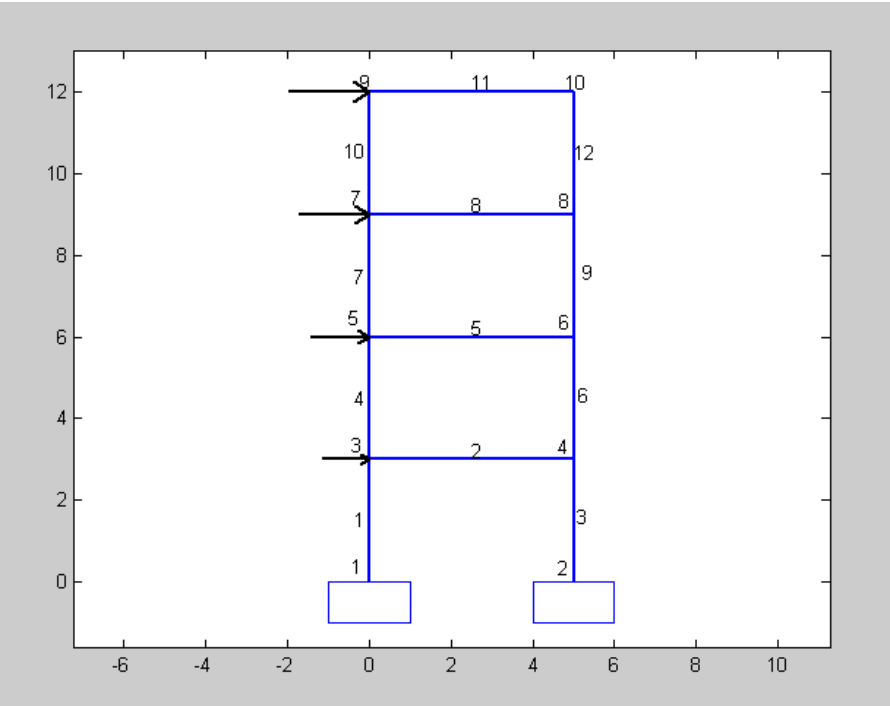

Figure 2 Model of four story frame

The forces applied are 20, 25, 30, 35 kn. The plastic moment at beam sections is 54kn*m.

$$\text{Mpl} = 1.5 \times Wy \times fy = 1.5 \times \frac{b \times h^2}{6} \times fy = 1.5 \times \frac{0.3 \times 0.4^2}{6} \times 4500 = 54kn \times m \tag{2}$$

After the analysis is finished we can read the data calculated by the program. There were a total of nine linear steps. Internal forces of the final step are shown in Figure 3. Diagram of internal moments and deformed shape for step 1,5,9 are shown in Figure 4, Figure 5, Figure 6. A plot of internal moments and deformed shape from a linear analysis is shown in Figure 7. A plot of force-displacement at node 3 is shown in Figure 8. A comparison of internal moments and deformed shape can be made between plastic analysis and linear analysis . This is shown in Figure 6 and Figure 7.

**********STEP NR-9**********

**********internal forces for elements from 1 to 12*********

ELEMENTS	{-------NODE(1)----------}			{--------NODE(2)---------}		
	axial N(1)	shear v(1)	moment M(1)	axial N(2)	shear V(2)	moment M(2)
element 1	-86.40	55.16	234.32	86.40	-55.16	-68.83
element 2	9.77	-21.60	-54.00	-9.77	21.60	-54.00
element 3	86.40	54.84	233.68	-86.40	-54.84	-69.17
element 4	-64.80	44.93	122.83	64.80	-44.93	11.96
element 5	12.59	-21.60	-54.00	-12.59	21.60	-54.00
element 6	64.80	45.07	123.17	-64.80	-45.07	12.04
element 7	-43.20	32.52	42.04	43.20	-32.52	55.51
element 8	14.98	-21.60	-54.00	-14.98	21.60	-54.00
element 9	43.20	32.48	41.96	-43.20	-32.48	55.49
element 10	-21.60	17.50	-1.51	21.60	-17.50	54.00
element 11	17.50	-21.60	-54.00	-17.50	21.60	-54.00
element 12	21.60	17.50	-1.49	-21.60	-17.50	54.00

Figure 3 Internal forces in the final step

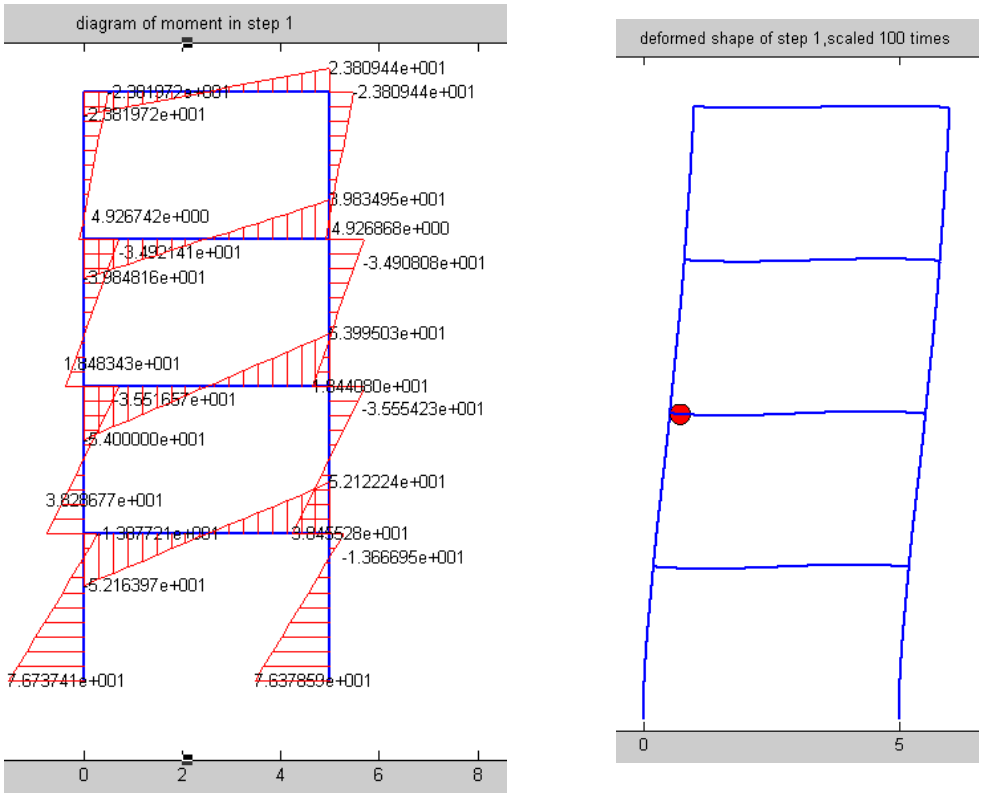

Figure 4 Diagram of internal moments and deformed shape in step 1

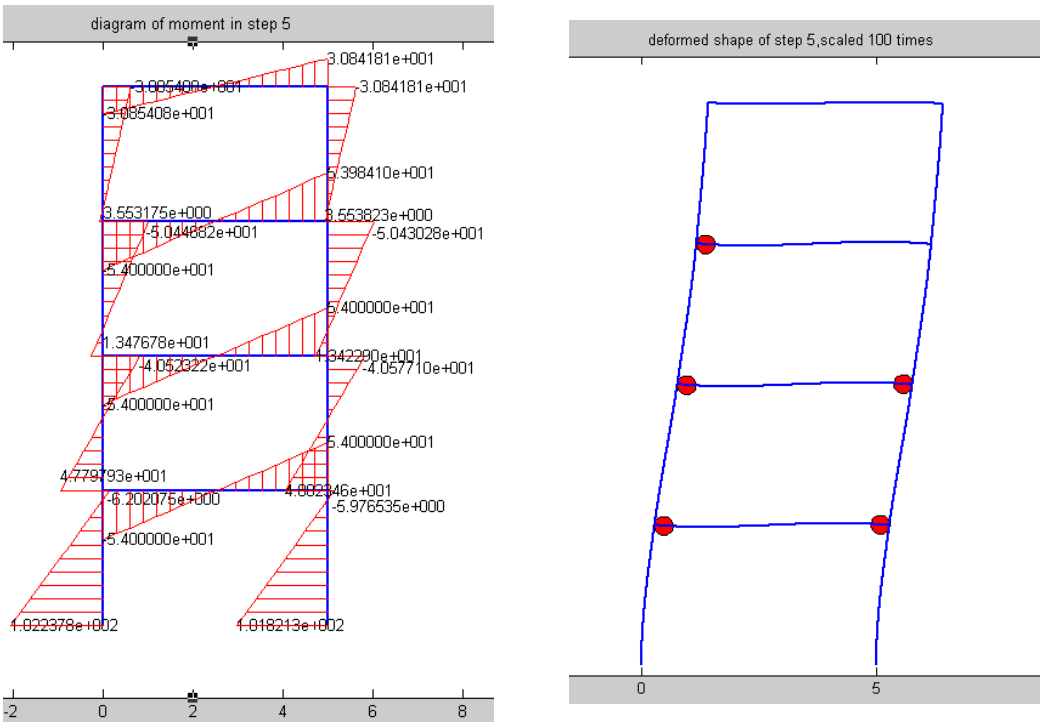

Figure 5 Diagram of internal moments and deformed shape in step 5

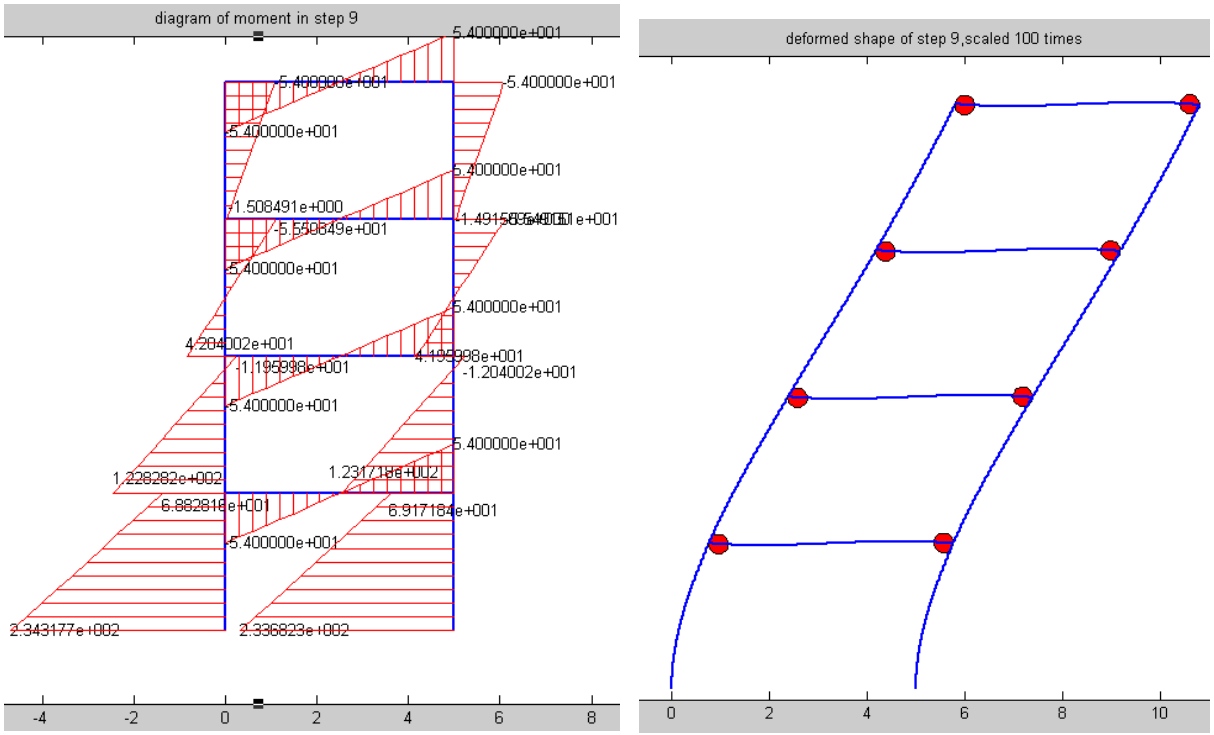

Figure 6 Diagram of internal moments and deformed shape in step 9 (final step)

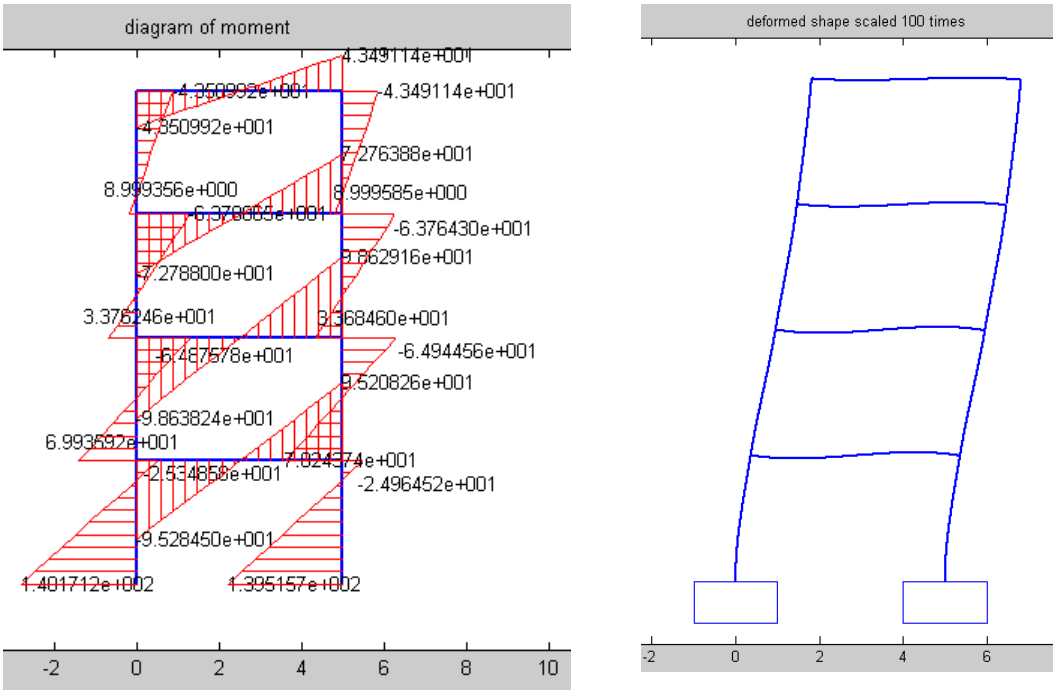

Figure 7 Internal moments and deformed shape from a static linear analysis

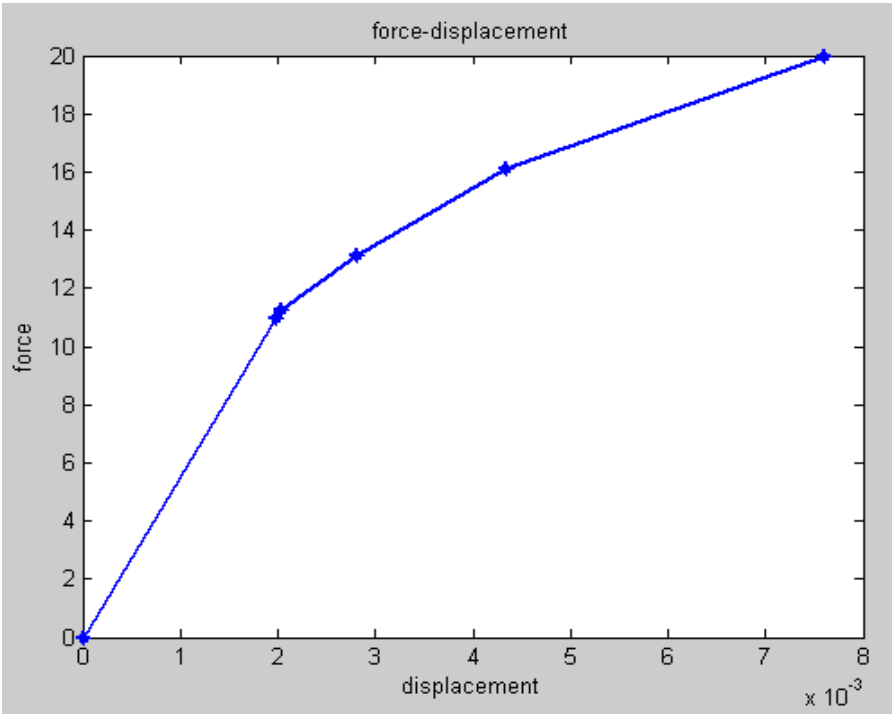

Figure 8 Force-Displacement at node 3

CONCLUSION

A step by step incremental linear analysis is used for the solution. Although only the M3 type plastic hinge is included, good results are obtained. The program can help in predicting the history of plastic hinge formation and redistribution of internal forces.

REFERENCES

[1] Niko Pojani, Niko Lako. Teoria e Strukturave DINAMIKA. *Shtepia botuese e librit universitar Tirane 2006*

[2] Niko Pojani, Hektor Cullufi, Niko Lako, Markel Baballeku, Rikard Luka. Metodat energjitike ne statiken e strukturave. Teoria dhe aplikime. *Botimet toena Tirane 2009*

[3] Niko Pojani. Inxhinieria sizmike. *Botimet toena Tirane 2003*

[4] Young W. Kwon, Hyochoong Bang. The finite element method using MATLAB *1997 by CRC press LLC*

[5] Andy H. Register A guide to MATLAB Object –Oriented Programming. *2007 by SciTech Publishing Inc.*

[6] Matlab Object –Oriented Programming. *1984-2011 by The MathWorks, Inc*.

[7] Matlab Graphics. *1984-2011 by The MathWorks, Inc.*