5th Youth Symposium on
Experimental Solid Mechanics

May, 10-13 2006
Púchov, Slovakia

COMPARATIVE ANALYSIS OF FEM SOFTWARE IN SOLVING DYNAMIC PROBLEMS

NENAD DJORDJEVIC[1], MIROSLAV ZIVKOVIC[2], SNEZANA VULOVIC[3], NENAD GRUJOVIC[4*]

[*] *Faculty of Mechanical Engineering University of Kragujevac; Kragujevac, Serbia and Montenegro*
[1] *e-mail: neshdjordjevic@yahoo.com*
[2] *e-mail: zile@kg.ac.yu*
[3] *e-mail: vsneza@kg.ac.yu*
[4] *e-mail: gruja@kg.ac.yu*

ABSTRACT: **There are many real structures where the loads are time-depend. Therefore we have to perform dynamic analysis to solve differential equations of motions, to meet transient response of model. Few effective methods for solving system of differential equations of motions are implemented in softwares which work on FEM (Finite element method) basis. Here is used direct Newmark step-by-step numerical time integration method, which is unconditionally stable.**
The objective of this work is to perform dynamic analysis of model in the software PAK (system of programs for structural analysis) [7] and compare those results with the results of the other FEM softwares. Agreements of the results would improve reliability of used software. Beside that, there are parameters and aspects of modeling which are considered here. All of this we have demonstrated and discussed on an example.

KEY WORDS: **FEM simulation, dynamic analysis, Newmark method**

1 INTRODUCTION

In the many practical examples, structures are subjected to time-varying excitation. Beside that, the inertial characteristics of structure have to be considered in analysis of motion. Therefore, we have to perform dynamic analysis to solve differential equations of motions, to compute transient response of model.
Two different numerical methods, which are implemented in FEM (finite element method) softwares, can be used for a transient response analysis: direct and modal. Direct method performs a numerical integration on the complete coupled equations of motions. The modal method utilizes mode shapes of the structure to reduce and uncouple equations of motions. Direct methods can be classified in two basic groups: implicit and explicit. Implicit methods are unconditionally stable, while stability of explicit methods depends of value of time-step increment [3]. Which method should be utilized depends upon structure and the nature of loading. Newmark step-by-step numerical time integration is used in this paper for computing the behaviour of a structure on time-varying excitation. Eigenvalues which correspond to natural vibrations and mode shapes are also computed here.
The methods, which are mentioned above, are approximate numerical methods, so the objective is to perform dynamic analysis of structure using PAK (system of programs for structural analysis) [7] and the other softwares which are based on finite element method. Agreements of the results would improve reliability of used softwares. We have also considered software's capabilities about dynamic analysis, setting up the type and parameters of analysis. All of this we have demonstrated and discussed on an example.

2 THEORETICAL BACKGROUND

2.1 Differential equations of motions

Model which is considered in this paper doesn't include damping effects, so the differential equation of motions, in this case, derived from principal of virtual work, is [6]:

$$M\ddot{U}+KU=F(t) \qquad (1)$$

where U, $\dot{U}$ and $\ddot{U}$ are vector of nodal displacement, velocity and acceleration, respectively, M is mass matrix and K is linear stiffness matrix. $F(t)$ is a vector of nodal external load.
In general, mass matrix and stiffness matrix are defined as:

$$M=\int_V \rho H^T H dV \qquad (2)$$

$$K=\int_V B^T C B dV \qquad (3)$$

where the ρ is density, H is matrix of interpolation functions, B is strain-displacement matrix and C is constitutive matrix.
For computing the response of the structures with great number of degree of freedom, it is useful to take lumped mass matrix formulation instead consistent mass matrix. Lumped mass matrix formulation contains only diagonal matrix-elements. Using the lumped mass matrix formulation reduces the time required for analysis.
Here are used standard and enhancement elements. Enhancement elements include incompatible displacements and thus the better results are obtained [10].

2.2 Integration of differential equations of motions

The implicit integration methods are unconditiable stable and theoretically can be counted with arbitrary time step. Differential equation (1) is satisfied at the end of time step, t+Δt, so it is written in form:

$$M\,{}^{t+\Delta t}\ddot{U}+K\,{}^{t+\Delta t}U={}^{t+\Delta t}F \qquad (4)$$

Newmark step-by-step numerical method substitute ${}^{t+\Delta t}\ddot{U}$ and ${}^{t+\Delta t}\dot{U}$ in above equation by

$${}^{t+\Delta t}\ddot{U}=\frac{1}{\alpha(\Delta t)^2}\left[{}^{t+\Delta t}U-{}^{t}U-{}^{t}\dot{U}\Delta t-\left(\frac{1}{2}-\alpha\right)(\Delta t)^2\,{}^{t}\ddot{U}\right] \qquad (5)$$

$${}^{t+\Delta t}\dot{U}=\frac{\delta}{\alpha\Delta t}\left({}^{t+\Delta t}U-{}^{t}U\right)-\left(\frac{\delta}{\alpha}-1\right){}^{t}\dot{U}-\left(\frac{\delta}{2\alpha}-1\right)\ \Delta t\,{}^{t}\ddot{U} \qquad (6)$$

to express unknown displacement ${}^{t+\Delta t}U$ as a function of known displacement, velocity and acceleration in time t [6]. Requested system of equations can be finally written as

$$\hat{K}\,{}^{t+\Delta t}U={}^{t+\Delta t}\hat{F} \qquad (7)$$

where matrix $\hat{K}$ and vector ${}^{t+\Delta t}\hat{F}$ are:

$$\hat{K}=K+a_0M \qquad (8)$$

$${}^{t+\Delta t}\hat{F}={}^{t+\Delta t}F+M\left(a_0\,{}^{t}U+a_2\,{}^{t}\dot{U}+a_3\,{}^{t}\ddot{U}\right) \qquad (9)$$

Coefficients a_0, a_2, a_3 are defined as

$$a_0=\frac{1}{\alpha(\Delta t)^2},\ a_3=\frac{1}{\alpha\Delta t},\ a_3=\frac{1}{2\alpha}-1 \qquad (10)$$

In this paper, Newmark parameters α and δ have a values

$$\delta=\frac{1}{2},\quad \alpha=\frac{1}{4} \qquad (11)$$

which correspond to accurate integration for linear change of acceleration [6].

2.3 Eigenvalues of the system

For determining critical time increment which will be used in the dynamic analysis, it is necessary to calculate eigenvalues. Eigenvalues which correspond to natural vibrations are calculated from:

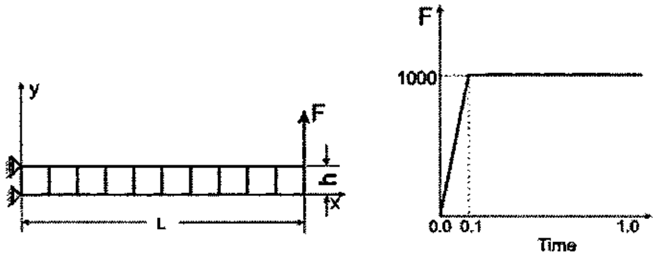
$$\left|\mathrm{K}-\omega_i^2\mathrm{M}\right|=0 \qquad (12)$$

where $\omega_i^2=\lambda_i$ is i-th eigenvalue. Eigenvalues are ordered like $\omega_1^2<\omega_2^2<...<\omega_n^2$.

3 EXAMPLE

3.1 Model description

Transient analysis and calculation of eigenvalues are performed for cantilever beam loaded on the free end (**Fig. 1**) [4]. Time-varyng excitation is also shown in **Fig. 1**.



$E=30\cdot10^6\dfrac{N}{cm^2}$

$A=0.1cm^2$

$h=1cm$

$L=10cm$

$\rho=0.3\dfrac{N}{cm^3}$

Fig. 1: Cantilever beam loaded on the free end

Model is generated using 2D four-nodes elements. Analysis is performed using FEM softwares: PAK, ADINA, COSMOS, ANSYS, NE Nastran and MSC Nastran.

3.2 Eigenvalues and mode shapes

In this example, numerical results for 10 eigenvalues are computed with v=0 and v=0.3 for consistent mass matrix and lumped matrix formulation. Results that are obtained by PAK are shown in **Tab. 1**.

Tab. 1: Numerical results for eigenvalues – PAK

MOD	Consistent		Lumped	
	v=0.	v=0.3	v=0.	v=0.3
1	16.076	16.058	15.954	15.936
2	99.356	98.602	94.534	93.875
3	250.260	250.170	246.260	242.710
4	274.310	269.860	249.740	249.840
5	529.250	514.840	442.360	433.070
6	756.960	756.180	665.710	647.270
7	859.940	828.590	743.080	741.580
8	1259.800	1200.700	900.350	871.640
9	1282.300	1284.600	1131.000	1091.700
10	1727.400	1638.300	1218.100	1210.600

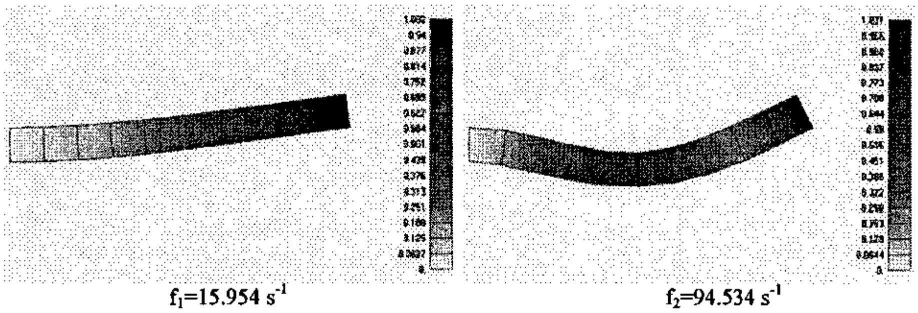
Poisson's ratio, v, impacts to the value of natural frequencies. The results that are obtained by PAK perfectly agree with the results that are obtained by the other softwares, as it is shown in **Tab. 2**.

Tab. 2: Numerical results for eigenvalues obtained by used softwares; lumped mass matrix, v=0

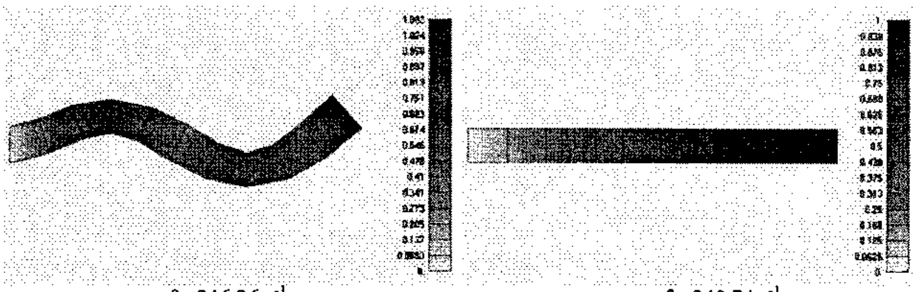
MOD	PAK	COSMOS	ANSYS	NE NASTRAN	MSC NASTRAN
1	15.954	15.954	15.954	15.954	15.954
2	94.534	94.534	94.534	94.535	94.535

MOD	PAK	COSMOS	ANSYS	NE NASTRAN	MSC NASTRAN
3	246.260	246.262	246.260	246.262	246.262
4	249.740	249.743	249.740	249.743	249.743
5	442.360	442.362	442.360	442.362	442.362
6	665.710	665.709	665.710	665.709	665.709
7	743.080	743.080	743.080	743.080	743.080
8	900.350	900.345	900.350	900.345	900.345
9	1131.000	1131.043	1131.000	1131.043	1131.040
10	1218.100	1218.119	1218.100	1218.119	1218.120

Mode shapes which correspond to the eigenvalues shown **Tab. 2** are shown in **Fig. 2**:



f_1=15.954 s^{-1} f_2=94.534 s^{-1}



f_3=246.26 s^{-1} f_4=249.74 s^{-1}

Fig. 2: Mode shapes of model with lumped mass matrix formulation and ν=0, PAK

The results that are displayed above are obtained using enhancement elements [6]. Eigenvalues which are obtained using standard elements by PAK and ADINA are identical, **Tab. 3**.

Tab. 3: Eigenvalues for the model which consists of standard elements – PAK

MOD	Consistent		Lumped	
	ν=0.	ν=0.3	ν=0.	ν=0.3
1	19.650	19.510	19.502	19.364
2	120.070	118.150	114.370	112.640
3	250.260	250.230	249.740	249.650
4	326.200	317.300	294.290	287.230
5	616.960	592.230	521.660	504.230
6	756.960	757.020	743.080	736.510
7	982.590	933.160	775.390	748.750
8	1282.300	1279.400	1037.800	988.460
9	1416.100	1337.600	1218.100	1206.300
10	1838.900	1783.200	1295.800	1235.900

Comparing the results shown in **Tab. 1** and **Tab. 3**, it can be noticed that standard elements (without enhancement) don't give correct results.

3.3 Transient response of cantilever beam

Transient response of the cantilever beam was analyzed in 50 time steps (increments) where the size of time step is $\Delta t = 0.006s$. Using Newmark step-by step numerical integration method with coefficients $\alpha=0.25$, $\delta=0.5$ (11), and enhancement elements, we have got good agreement of results for both matrix formulations, **Fig. 3,4.**

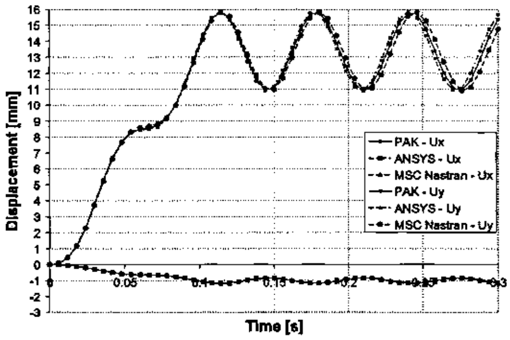


Fig. 3: Displacement in the loaded node, v=0; Consistent mass matrix formulation

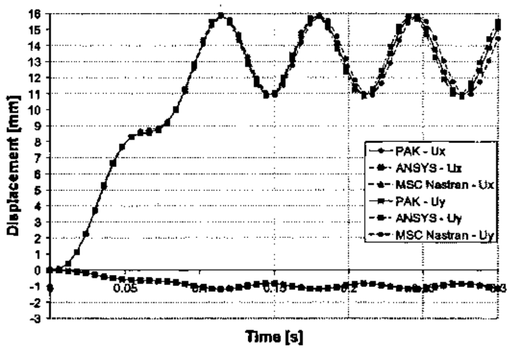


Fig. 4: Displacement in the loaded node, v=0; Lumped mass matrix formulation

It can be noticed that the curve which corresponds to results obtained by PAK is just between ANSYS and MSC Nastran curves.
COSMOS performs only modal superposition method and does not support direct integration method. Results obtained by COSMOS and PAK are shown in **Fig. 5.**

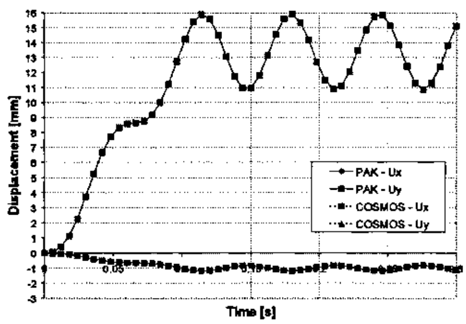


Fig. 5: Displacement in the loaded node; COSMOS and PAK

Even two softwares are based on different methodologies, there is very good agreement.

We have also compared the results obtained using standard elements in PAK and ADINA. The results are identical, **Fig. 6**.

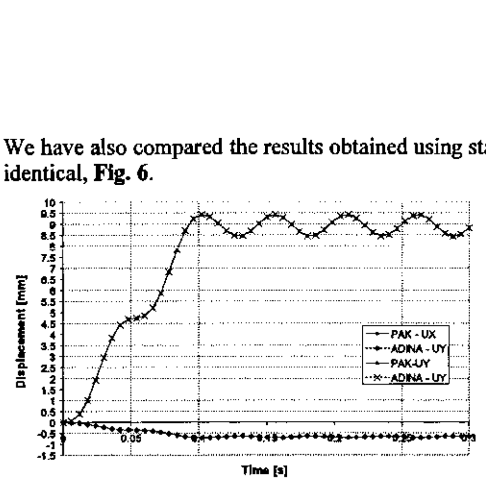


Fig. 6: Standard elements ADINA and PAK

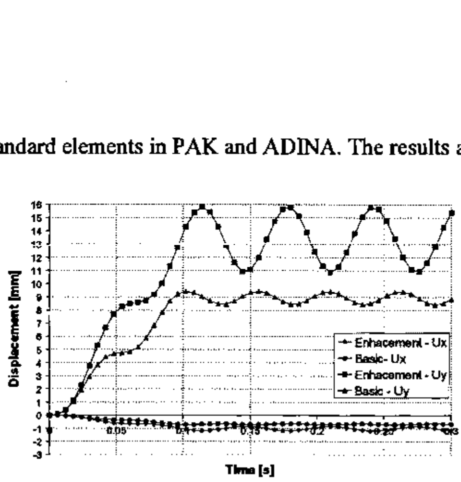


Fig. 7: Enhancement and standard elements PAK

Comparing the standard and enhancement elements in PAK, **Fig. 7**, improves the fact that the standard element has a greater stiffness. Maximal displacement is almost doubly less in relation to results with enhancement elements while the amplitude of vibrations is significantly smaller.

4 CONCLUSION

Developing of FE methods have afforded the efficient solving of real dynamic problems. Transient response analysis is the most general method for computing forced dynamic response. Modal transient response is an alternate approach to computing the transient response of structure, but it is approximation because it uses only n natural frequencies. Calculating the eigenvalues is used for determining the size of time increment, which would be used in dynamic analysis.

Direct integration method, which is implemented in software PAK, provides the reliable results for the models with enhancement element (field of incompatible displacement is added), but not for the standard elements. Example improves the fact that standard elements behave like that stiffness is greater. Excellent agreement of results is shown in calculation of eigenvalues (natural frequencies).

At least, according to the shown results in this paper, it should be realized that numerical methods for computing dynamic response, implemented in the FE softwares, provide acceptable agreements of results.

5 REFERENCES

[1] ADINA: *A Finite Element Program for Automatic Dynamic Incremental Nonlinear Analysis*, ADINA Engineering Inc, Watertown, USA 1984.

[2] ANSYSDOCUMENTS: Ansys Inc, Southpointe

[3] BATHE, K.J.: *Finite Element Procedures in Engineering Analysis*, Prentice-Hall Civil Engineering and Engineering Mechanics Series, New Jeresy USA

[4] COSMOS/M Advanced Dynamic, Structural Research and Analysis Corporation (SARC), 2001.

[5] FEMAP User Guide, Unigraphic Solution Inc, Exton, USA 2002.

[6] KOJIC, M., SLAVKOVIC, R., ZIVKOVIC, M., GRUJOVIC, N.: *Finite Element Method - linear analaysis*, Faculty of Mechanical Engineering University of Kragujevac, Kragujevac, SCG, 1998

[7] KOJIC, M., SLAVKOVIC, R., ZIVKOVIC, M., GRUJOVIC, N.: *PAK – S Program for FE Structural Analysis USER MANUAL*, Faculty of Mechanical Engeneering, Kragujevac

[8] MSC Nastran, *Dynamic Analysis*, http://www.mscsoftware.com/support/online_ex/

[9] NE/Nastran User's Manual Version 8.1, Noran Engineering Inc, Los Alamitos, USA 2002

[10] SLAVKOVIC R., ZIVKOVIC M., KOJIC M.: *Enhanced 8-node three-dimensional solid and 4-node shell elements with incompatible generalized displacement*, Comunication of numerical methods in engineering, Vol. 10, 699-709, 1994