10.2478/tvsb-2014-0023

Transactions of the VŠB – Technical University of Ostrava

No. 2, 2014, Vol. 14, Civil Engineering Series

paper #23

Maciej MAJOR[1], Izabela MAJOR[2]

COMPUTER AIDED DESIGN – COMPARATIVE ANALYSIS OF WIDELY AVAILABLE SOFTWARE WITH ANALYTICAL METHOD

Abstract

In this article a comparative analysis was performed between analytical method and widely available computer programs employed in beam design like: ADINA, Robot Structural Analysis, Intersoft R2D2 and RM-WIN. In the analysis a simple case of a simply supported beam and uniformly distributed load over the length of the span was assumed. The conclusions from analysis are presented at the end of the study.

Keywords

Aided design, beam, computer applications.

1 INTRODUCTION

Computer structures design is an element which makes structure designing easier and faster. Nowadays an engineer does not need to do laborious calculations with help of calculator because computer software can be used. In this article the vertical displacement problem is presented by means of comparing analytical method and widely used and available beam design software. The comparison concerns a simple case of a simply supported beam with continuous load over the span length. For two dimension analysis the following programs ADINA, Autodesk Robot Structural Analysis, Intersoft R2D2 and RM-WIN are used. The first two applications enabled three dimensional analysis (3D).

ADINA – Automatic Dynamic Incremental Nonlinear Analysis – is a tool which enables mainly tension and solid body deformations analysis. This software also allows to analyze fluid and gases flow velocity, pressure, tension and deformation of environment where fluid or gas is being set (for example pipes) [1]. Moreover analysis of linear or nonlinear material [2,3], dynamics, thermal conductivity and wave propagation can be done and many other calculations. Software environment allows to work both on two and three dimensions.

For calculations only Finite Element Method is used [4,5]. This program does not have a predefined profiles module or endurance analysis module – an engineer needs to estimate whether the given structure will be enough enduring.

Autodesk ROBOT Structural Analysis – is a tool commonly used by constructor engineers. This software has extensive interface which allows to analyze two and three dimensional beam, plate (see [6]) and solid blocks. The software environment also provides statics, dynamics, modal, construction efforts, linear or non-linear materials analysis by means of finite element method. There are many other modules easily calculating and saving time such as typical trusses, frames, surfaces or solid structures, foundations, beams, reinforced concrete slabs, parameterized structures and more.

[1] Maciej Major, Ph.D., Faculty of Civil Engineering, Czestochowa University of Technology, ul. Akademicka 3, 42-200 Częstochowa, Poland, e-mail: mmajor@pcz.czest.pl.

[2] Izabela Major, Ph.D., Faculty of Civil Engineering, Czestochowa University of Technology, ul. Akademicka 3, 42-200 Częstochowa, Poland, e-mail: admin@major.strefa.pl.

The program is not complicated but requires some practice and experience from a user to design complex structures.

Intersoft R2D2 – program for any static beam scheme analyzing only in two dimension environment. With this tool the following analysis can be taken: statics analysis of construction or beams endurance under load. It is worth noticing that beams endurance analysis requires installing additional modules - it is not enough to have a standard version of the program to provide this type of calculations.

RM-WIN – it is also an application for analyzing beam in two-dimensional environment (2D). With that program any static beam scheme and also beam endurance can be analyzed. That application is the easiest to control of all described programs in this article. Moreover RM-WIN is the only program that has free of charge module for non-commercial usage.

2 CALCULATION ASSUMPTIONS

For the analysis a naturally supported steel beam with a square cross-section was chosen. Over the span length there was continuous load, noted as q (Fig. 1).The following material data (Young modulus and Poisson's ratio) were implemented for steel:

$$E = 210\,\text{GPa}$$
$$\upsilon = 0.30 \tag{1}$$

dimensions and beam load

$$L = 4.00\,\text{m}$$
$$a = 0.10\,\text{m} \tag{2}$$
$$b = 0.10\,\text{m}$$
$$q = 2.00\,\text{kN/m}$$

The beam's self-weight is omitted in further considerations.

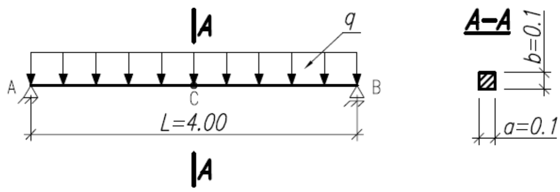

Fig. 1: Geometry of the beam being analyzed. All dimensions are given in [m]

3 ANALYTICAL METHOD

In the analytical calculations Clebsch method is used. Reactions in both beam ends are (see [7])

$$V_A = V_B = \frac{qL}{2} \tag{3}$$

The differential equation has the form of

$$EJw'' = -M(x) \tag{4}$$

where

$$M(x) = V_A \cdot x - q \cdot x \cdot \frac{x}{2} \tag{5}$$

having substituted (5) to (4) we obtain

$$EJw'' = -V_A \cdot x + \frac{qx^2}{2} \tag{6}$$

Having integrated twice we get

$$EJw' = C - \frac{V_A x^2}{2} + \frac{qx^3}{6}$$

$$EJw = D + Cx - \frac{V_A x^3}{6} + \frac{qx^4}{24} \tag{7}$$

Integral constants are determined from boundary conditions (after substituting (3))

$$w(0) = 0 \quad \rightarrow \quad D = 0$$

$$w(L) = 0 \quad \rightarrow \quad C = \frac{qL^3}{24} \tag{8}$$

Taking into account integral constants in equation (8) and $x = L/2$ in equation (7) we obtain final beam displacement equation as

$$w = \frac{5qL^4}{384EJ} \tag{9}$$

By substituting the given value and numerical values presented at (1) and (2) and $J = ab^3/12$, into the above equation (9) the mid-span vertical displacement value is obtained (point C at Fig. 1)

$$w = 0.00381\,m \tag{10}$$

4 COMPUTER AIDED DESIGN

At first in two dimension environment (2D) a simply supported beam under a uniform load was modeled. At Fig. 2 final results for ADINA program are shown. The extreme value of vertical displacement for the mentioned beam is at 0.003810m.

Fig. 2: Simply supported beam in 2D environment – ADINA program

The second examined program is Autodesk ROBOT Structural Analysis. In this case the extreme value of mid-span vertical displacement is 0.00381m (Fig. 3)

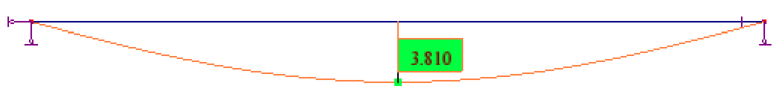

Fig. 3: Simply supported beam in 2D environment – Autodesk ROBOT Structural Analysis program

Having declared a beam in Intersoft R2D2 program, the value of vertical displacement given by the program is identical to the previous two examples (Fig. 4)

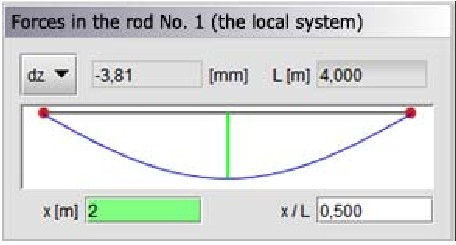

Fig. 4: Simply supported beam in 2D environment – Intersoft R2D2 program

The last application which was used for two-dimensional (2D) analysis is RM-WIN. As it was expected also in this case the value of displacement is 0.0038m (Fig. 5).

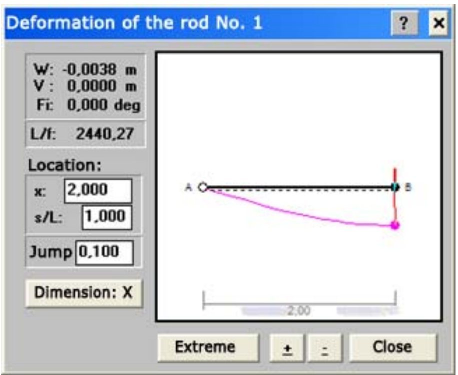

Fig. 5: Simply supported beam in 2D environment – RM-WIN program

Fig. 6: Spatially modeled beam with continuous load (surface divided in half at length dimension) – ADINA program

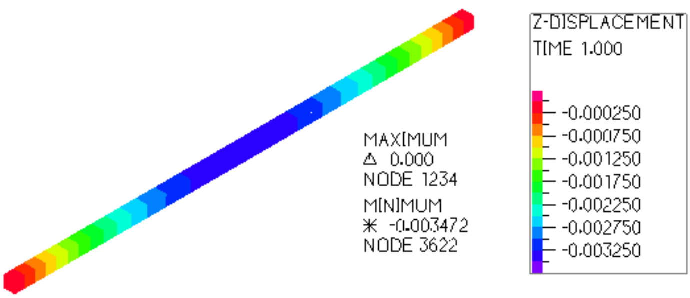

Fig. 7: Spatially modeled beam with pressure load at the top surface – ADINA program

The three-dimensional analysis was done with ADINA and ROBOT application. In ADINA application two types of loading (Fig. 6 and Fig. 7) were taken into account and the extreme value of vertical displacement equal to 0.003472m was obtained.

In Autodesk ROBOT Structural Analysis results were obtained for a beam declared as solid block (Fig. 8). The mid-span vertical displacement was 0.003801m.

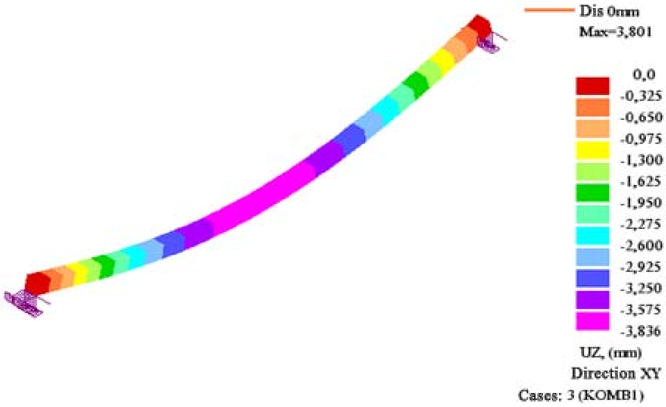

Fig. 8: Spatially modeled beam – ROBOT program

In ROBOT program there is a possibility to define a computation model as panel (surface). In such a scheme the results of vertical displacement are shown in Fig. 9. In this case the extreme value of vertical displacement equals 0.003816m.

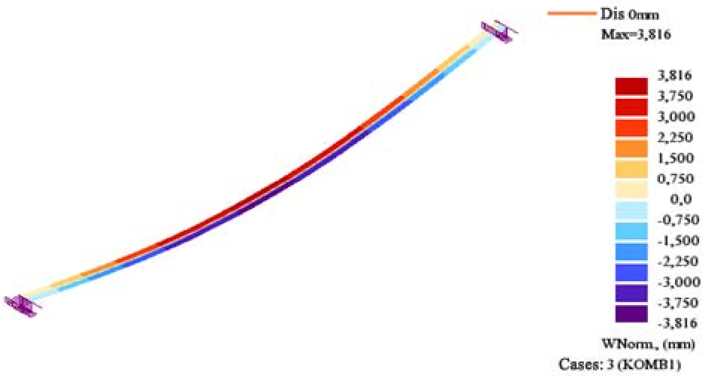

Fig. 9: Result for panel (surface) in ROBOT program. Positive and negative values are associated with local coordinate system for each panel

5 CONCLUSIONS

The analysis carried out in the above mentioned computer programs for simply supported beam with continuous load prove that beam modeled in two dimensional environment have identical results – vertical displacement values correspond the values obtained by analytical method. In case of finite element method used to calculate two dimension beam in ADINA program the vertical displacement value was equal to that achieved by analytical method. In this case the beam was divided into 0.02m sections each. With higher density of finite element on the net the results do not change, but in case of a less density net some discrepancy in results may appear. Summing up it can be stated that in case of two dimension (2D) analysis of vertical displacement in all four programs the results are the same and are equal to those in the analytical method. In case of three dimension analysis in ADINA program (Fig. 6 and Fig. 7) the differences are at tenth of millimeter and the error compared to analytical calculations have the value of

$$Err_{ADINA} = \left| \frac{0.003810 - 0.003472}{0.003809} \right| = 8.87\% \tag{11}$$

In case of ROBOT program (Fig. 8 and Fig. 9) the differences in results appear at hundredth of millimeter and the errors compared to analytical calculations are:

- for solid block

$$Err_{ROBOT} = \left| \frac{0.003810 - 0.003801}{0.003810} \right| = 0.24\% \tag{12}$$

- for panel (surface)

$$Err_{ROBOT} = \left| \frac{0.003810 - 0.003816}{0.003810} \right| = 0.16\% \tag{13}$$

While there is conformity of results in two dimensional (2D) beam design with analytical calculations then in three dimensional (3D) design some discrepancy has occurred. Finite element method is currently the most commonly used one and most helpful to solve some complex design problems with, especially when it is either impossible to make use of other methods or time-consuming or burdened with the error risk. On the basis of the comparative analysis it can be seen that Autodesk ROBOT Structural Analysis managed much better than ADINA despite the fact that the same MES net in both examples was set (cubes with 0.02m long sides). It is important that applying the net consisting of tetrahedrons makes a structure more rigid and the results are far different from the values obtained in the analytical method. Summing up it can be stated that the indisputable advantage of program calculations is its high efficiency involving significant time reduction for simple and complex structures. However, proper calculations depend on the program being used and the engineer's skills.

REFERENCES

[1] BATHE, K.J., LARSSON, G. The ADINA system in engineering practice, *Finite Elements in Analysis and Design*, Volume 2, Issues 1–2, April 1986, Pages 41–60, ISSN: 0168874X

[2] HARIRIAN, M., CARDOSO, J.B., & ARORA, J.S. Use of ADINA for design optimization of nonlinear structures, *Computers and Structures*, Volume 26, Issue 1-2, 1987, Pages 123-133, ISSN: 00457949.

[3] MAJOR, M. *Modelowanie zjawisk falowych w hipersprężystym materiale Zahorskiego*, Wyd. Politechniki Częstochowskiej, Częstochowa 2013, 188 p. ISBN 978-83-7193-600-5.

[4] BATHE, K.J. *Nonlinear finite element analysis and ADINA*. Mechanics Research Communications, Volume 12, Issue 6, November 1985, Page 346, ISSN: 00936413.

[5] BATHE, K.J. Finite elements in CAD and ADINA, *Nuclear Engineering and Design*, Volume 98, Issue 1, 1986, pp. 57-67, ISSN: 00295493

[6] KURZAK, L., MAJOR, M. & MAJOR, I. *Computer-Aided Design Using Programs ROBOT and ADINA - Comparison.* LEHOCKA, D., CARACH, J., KNAPCIKOVA, L., HLOCH, S. In: 5th International Scientific and Expert Conference of the International TEAM Society. Presov: 2013, pp. 146-149, ISSN 1847-9065.

[7] MAJOR, M. and MAJOR, I. *Zagadnienia z mechaniki ogólnej rozwiązywane MATHCAD-em. Cz. I. Statyka*. Wyd. P.Cz., Częstochowa 2005, 109 p. ISBN 83-7193-281-2.

Reviewers:

Ing. Mikolášek David, Ph.D., Department of Structural Mechanics, Faculty of Civil Engineering, VŠB-Technical University of Ostrava, Czech Republic.

Ing. Aleš Nevařil, Ph.D., Institute of Structural Mechanics, Faculty of Civil Engineering, Brno University of Technology, Czech Republic.