УДК 624.075

МОДЕЛЮВАННЯ СКІНЧЕННО-ЕЛЕМЕНТАРНОЇ МОДЕЛІ КОНСТРУКЦІЇ, ВИКОРИСТОВУЮЧИ ПРОГРАМНИЙ КОМПЛЕКС ANSYS

FINITE ELEMENT ANALYSIS SIMULATION OF ASSEMBLY USING ANSYS

Білокуров П.С., к.т.н., доцент (Національний авіаційний університет, м.Київ)

P.S. Bilokurov., Ph. D, Candidate in Engineering (National Aviation University, Kyiv)

В статті приведені основновні положення для разрахунку скінченно-елементарної моделі конструкції споруди в програмному комплексі ANSYS

The article deals with the question of the procedures of constructing an ANSYS nonlinear finite element model of building element.
The structural integrity and stability of any building is only as good as its individual parts. The way those parts fit together, along with the choice of materials and the building site, all contribute to how the building will perform under normal — or extreme — conditions. Civil engineers integrate this knowledge into their building designs and comply with increasingly demanding safety and government regulations. At the same time, the general public is demanding environmentally conscious designs.
ANSYS simulation software gives designers the ability to assess the influence of these variables in a virtual environment.
Through visualizing the effect of a wide range of variables, engineers can narrow the scope of field investigations, save considerable time and cost on projects, and move more quickly to the groundbreaking stage.
Civil engineers use ANSYS for projects as diverse as high-rise buildings, bridges, dams, stadiums, etc. By experimenting with innovative design in a virtual environment, dams, stadiums, etc. By experimenting with innovative design in a virtual environment, engineers and designers can analyze safety, strength, comfort and environmental considerations. Ключові слова:

Ключові слова :
Моделювання, скінченно-елементарна модель, будівельні конструкції, міцність та деформативність
Reinforced concrete, deformability, strength, building elements, modelling finite elements, ANSYS

Inroduction. Most finite element simulations assume that all the elements begin in a stress, strain and deflection free state. However, in reality, most structures, especially those that involve construction, will have residual stresses and strains caused by the assembly sequence.

Finite element method (FEM) models were developed to simulate the behavior of four full-size beams from linear through nonlinear response and up to failure, using the ANSYS program (ANSYS 1998). Comparisons were made for load-strain plots at selected locations on the beams; load-deflection plots at midspan; first cracking loads; loads at failure; and crack patterns at failure[5,6].

The models were subsequently expanded to encompass the linear behavior of the Horsetail Creek Bridge. Modeling simplifications and assumptions developed during this research are presented. The study compared strains from the FEM analysis with measured strains from load tests. Conclusions from the current research efforts and recommendations for future studies are included.

The Finite Element Method (FEM) involves dividing the complex domain into finite elements and uses variational concepts to construct an approximation of the solution. There are two types of analysis: 2-D modeling and 3-D modeling. A 2-D modeling is simple, can be run on normal computers but may give less accurate results on some applications[1].

However, a 3-D modeling produces more accurate results while sacrificing the ability to run effectively on all but the fastest computers. Within each of these modeling schemes, numerous algorithms (functions) can be inserted to make the system behave linearly or non-linearly. Linear systems are far less complex and generally do not take into account plastic deformation. Non-linear systems do account for plastic deformation, and many also are capable of analyze a material all the way to fracture.

Literature Review. The finite element analysis can be traced back to the work by Alexander Hrennikoff (1941) and Richard Courant (1942). Hrenikoff introduced the framework method, in which a plane elastic medium was represented as collections of bars and beams. These pioneers share one essential characteristic: mesh discretization of a continuous domain into a set of discrete sub-domains, usually called elements.

• In 1950s, solution of large number of simultaneous equations became possible because of the
digital computer.

• In 1960, Ray W. Clough first published a paper using term “Finite Element Method”.

• In 1965, First conference on “finite elements” was held.

• In 1967, the first book on the "Finite Element Method" was published by Zienkiewicz and Chung.

• In the late 1960s and early 1970s, the FEM was applied to a wide variety of engineering problems.

• In the 1970s, most commercial FEM software packages (ABAQUS, NASTRAN, ANSYS, etc.) originated. Interactive FE programs on supercomputer lead to rapid growth of CAD systems.

• In the 1980s, algorithm on electromagnetic applications, fluid flow and thermal analysis were developed with the use of FE program.

Research significance and objectives. In the given article there are considered the next problem:

- analysis of ANSYS simulation software for designing of building structures;
- computer modeling of building elements using ANSYS.

Background. The ANSYS program is capable of simulating problems in a wide range of engineering disciplines [4].

Structural Analysis: Deformation, stress, and strain fields, as well as reaction forces in a solid body.

Thermal Analysis: Steady-state or time-dependent temperature field and heat flux in a solid body.

Structural Analysis

This analysis type addresses several different structural problems, for example:

Static Analysis: The applied loads and support conditions of the solid body do not change with time.

Nonlinear material and geometrical properties such as plasticity, contact, creep, etc., are available.

Modal Analysis: This option concerns natural frequencies and modal shapes of a structure.

Harmonic Analysis: The response of a structure subjected to loads only exhibiting sinusoidal

behaviour in time.

Transient Dynamic: The response of a structure subjected to loads with arbitrary behaviour in time.

This software is a suite of powerful engineering simulation programs, based on finite element method, which can solve problems ranging from relatively simpler linear analyses to the most challenging non-linear simulations. The analysis of a structure with ANSYS is performed in three stages:

a) Pre-processing
b) Analysis solver
c) Post-processing

Finite Element Analysis Procedure (Structures)

• Pre-processing
• Discretization of the structure – Meshing
• Assign element type and properties
• Assign material properties
• Apply Boundary conditions and Loads

• Solution
• Select the solver
• Calculate element stiffness matrices
• Assemble global stiffness matrix
• Solve for displacements, strains, stresses etc.
• Post-processing
• Display / Output displacements, strains, stresses etc.
• Calculate user defined parameters from the results

Some helpful hints for modeling construction sequences:

It is often necessary to fix displacements of elements in their killed state since they are still active in the analysis. Fixing nodal degrees of freedom of killed elements will prevent excessive deformation caused by either the displacements of the surrounding elements or free rigid body motion. If you add these artificial constraints, be sure to remove them in a manner such that artificial stresses are not induced in the model upon activation [2].

Annealing can be simulated by simply killing and then reactivating elements in subsequent load steps.

If contact elements are attached to the regions being killed, be sure to also kill the contact pair.

Activation or deactivation generally occurs instantaneously. This stepped change nonlinearity (similar to changing contact status) can cause convergence issues when a large number of elements are being killed or reactivated. Activation/Deactivation of small sections at a time will minimize these problems.

For materials whose "born" stiffness needs to be gradually ramped on, temperature dependent material properties in conjunction with dummy temperature loads can often be used to simulate this gradual process.

When postprocessing, be careful to deactivate the killed elements to avoid unrealistic results. Inaccurate stress results, in particular, can occur if the killed element stresses (which are set to zero) are averaged with active elements [3].

As an initial step, a finite element analysis requires meshing of the model. In other words, the model is divided into a number of small elements, and after loading, stress and strain are calculated at integration points of these small elements (Bathe 1996). An important step in finite element modeling is the selection of the mesh density. A convergence of results is obtained when an adequate number of elements is used in a model. This is practically achieved when an increase in the mesh density has a negligible effect on the results (Adams and Askenazi 1998). Therefore, in this finite element modeling study a convergence study was carried out to determine an appropriate mesh density.

Recommended FE modeling procedure for reinforced concrete beams

1. The symmetry of the beams should be used to reduce computational time and computer disk space requirements. In this project, a quarter of the full-size beam, with proper boundary conditions, was used for modeling.

2. A steel plate needs to be included in the models at the support locations to represent the actual support condition in the full-size beams. The steel plate also provides a more even stress distribution over the support area to avoid problems of stress concentration.

3. For nonlinear analysis of a reinforced concrete beam, the total load applied to a model must be divided into a number of load steps. Sufficiently small load step sizes are required, particularly at changes in behavior of the reinforced concrete beam, i.e., major cracking of concrete, yielding of steel, and approaching failure of the reinforced concrete beam. Properly defining minimum and maximum sizes for each load step, depending upon the behavior of the reinforced concrete beam, assists in convergence of the solutions and reduces computer computational time.

The use of ANSYS software is very good to know the process of collapse a reinforced concrete beam flexural cracks start to the shear cracks (linear), but the result is having a significant deviation in the phase of destruction of concrete (plastic). However, this shortcoming can be overcome by using multilinear plasticity material models available in ANSYS. Results of calculations are give on Fig.1. and Fig.2.

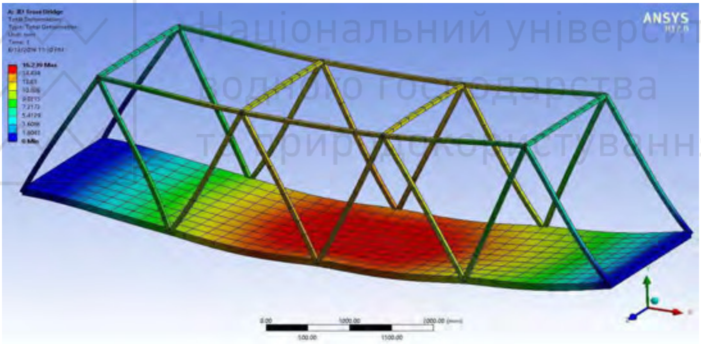

Figure 1. Stress Contour of the Bridge Elements under Tensile Collapsed Mechanism

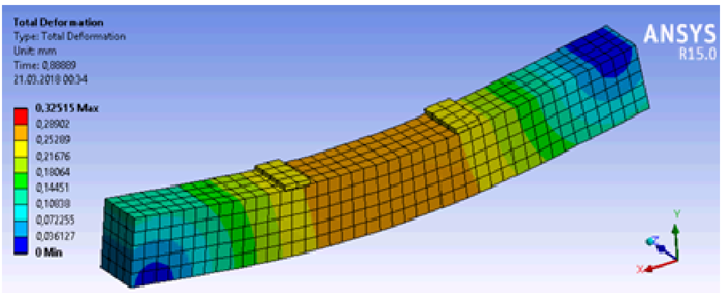

Figure 2. Stress Contour of the Beam under Compressive Collapsed Mechanism

ANSYS offers a comprehensive software suite that spans the entire range of physics, providing access to virtually any type of engineering simulation required in the design process. Organizations around the world trust ANSYS FEA and CFD software to deliver the best value for their engineering simulation software investment.

Conclusions. Finite element analysis (FEA) tools from ANSYS provide the ability to simulate every structural aspect of a product:

Linear static analysis that simply provides stresses or deformations

Modal analysis that determines vibration characteristics.

Advanced transient nonlinear phenomena involving dynamic effects and complex behaviour ansys-vibration-analysisAll users, from designers to advanced experts, can benefit from ANSYS structural analysis software through: the renowned ANSYS Workbench intuitive workflow; wide variety of material models; element library quality; solver speed, robustness and accuracy; and the ability to model every product — from single parts to very complex assemblies with hundreds of components interacting through contacts or relative motions. The speed of ANSYS simulations can be reduced further through efficient parallel processing capabilities to exploit multiple core PCs and High Performance Computing (HPC).

1. ACI Committee 318 (2007), "Building Code Requirements for Reinforced concrete (ACI 318-2007) and Commentary (ACI 318R-2007)", American Concrete Institute, Detroit, USA.

2. ANSYS Inc. (2006), "ANSYS Multiphysics-Finite Element Analysis SoftwareVersion 11.0" Canonsburg, Pennsylvania, USA. 7. ANSYS (2006), ANSYS user's Manual Revision 11.0, ANSYS Inc., Canonsburg, Pennsylvania, USA.

3. Vasudevan and Kothandaraman, G.(2011), “Parametric study on Nonlinear Finite Element Analysis on flexural behaviour of RC beams using ANSYS”, International Journal of Civil and Structural Engineering,2(1),pp-98-111

4. Neha, S. and Malipatil, M. (2014), “Parametric Study on Reinforced Concrete Beam using ANSYS”, Civil and Environmental Research,6(8),pp-88-94

5. Patil, S. and Niranjan, R. (2012), “Non Linear Finite Element Method of Analysis of Reinforced Concrete Deep Beam” International Journal of Modern Engineering Research,2 (6),pp-4622-4628.

6. N. Sundar, P. N. Raghunath and G. Dhinakaran, Flexural Behavior of RC beams with Hybrid FRP Strengthening. International Journal of Civil Engineering and Technology, 7(6), 2016, pp.427–433.