Methodology of teaching methods of FEM analysis using software ANSYS

Juraj Králik, Juraj Králik JR.,
University of Technology in Bratislava,
Faculty of Civil Engineering and Faculty of Architecture and Design,
Vazovova 5, Bratislava 812 43
Slovakia

Received: April 12, 2021. Revised: October 15, 2021. Accepted: December 5, 2021. Published: January 4, 2022.

Abstract—This paper presents the authors' experience of teaching the finite element method (FEM) at university. With the development of computational tools in the second half of the twentieth century, there was also the development of computational methods focused on the algorithmization of engineering tasks based on FEM. From the solution of individual problems of the state of stress and deformation from the influence of the external environment, a complex solution of the mutual interaction of the system of deformable bodies (elements) has been performed while improving the physical and geometric characteristics of modern materials and structures. Many processes in the automatic design system take place as if in a "black box" and the process of verifying the achieved results becomes the most important stage in the design activity. Without knowledge of the theoretical basis of FEM, physical and mathematical modeling, verification procedures and methods, the design of a structure cannot be safe and reliable. In this paper we present one of the possibilities how the student can get acquainted with the theoretical foundations of FEM and with computational procedures using ANSYS software.

Keywords—FEM, APDL, ANSYS, Statics, Stability, Dynamics, Nonlinearity, Structures, Safety, Control.

I. INTRODUCTION

The emergence of finite approaches to solving engineering problems can be dated back to 1906, when K. Weighard [1] proposed a solution to the state of stress using the so-called grid analogy (network analogy). The elastic continuum was modeled by a system of the same elastic elements (members), where the properties of the members were chosen so that the displacements at their joints (now nodes) have approximately

the same value as the displacements of points at these points of the elastic continuum. The work of L. F. Richardson (1910), [2], who analyzed the state of tension of the Aswam Dam dam by the method of networks and R.V. Soutwell (1946), [2], who contributed to the development of an iterative relaxation method used to solve large systems of algebraic equations. It was not until 1943, R. Courant [3], who was the first to propose the finite element method as it is to this day. In the finite analysis, Courant proposed to use the principle of stationary potential energy and interpolation functions of triangular sub - regions using polynomials.

On the other hand, J.W. Rayleigh, W. Ritz, and B.G. Galerkin [2, 4] demonstrated the recognition that continuum mechanics problems (one -, two - and three - dimensional problems) can be solved by applying a minimum of potential energy. or equivalent functionals. B. Galerkin [2] convert a continuous operator problem, such as a differential equation, commonly in a weak formulation, to a discrete problem by applying linear constraints determined by finite sets of basis functions. Often when referring to a Galerkin method, one also gives the name along with typical assumptions and approximation methods used as follows: Ritz–Galerkin method, Bubnov–Galerkin method and Petrov–Galerkin method. In 1953 S. Levy [2] formulated the stiffness equations in matrix form so that it was possible to use computer technology.

The first practical calculations by electronic computers were performed in the 1950s in the USA for aeronautical calculations (aircraft, missiles), M. J. Turner et al (1956) and J.H. Argyris (1960) [5]. The concept of finite element in calculations in 1960 was introduced by R. W. Clough (1960), [6], to emphasize the difference between finite and differential elements (differences). Further increase in the efficiency of computer technology in the sixties of the 20th century caused a rapid development of FEM - new types of elements, analysis of shells and plates as well as the application of FEM in new, so-called. non-structural areas, e.g. heat dissipation and others. Further development of the FEM at that time encountered

difficulties related to solving large systems of simultaneous equations, and so the development of the FEM had to wait for the development of electronic automatic computers and the creation of highly efficient programming languages.

In the period (1970 - 1980) supercomputers (BCP, CYBER 205, CRAY) were created with a speed of 10 MFLOPS, a memory of 4 Mwords 64 bits. In this period, large software packages, such as ANSYS and ABAQUS and others, were created for linear as well as nonlinear analysis of structures and materials. At the beginning of the period 1980 - 1990, a new phenomenon approached the development and application of FEM - a microcomputer (PC - personal computer). Significant PC calculation parameters allow the installation of FEM programs on these computers. The current development of graphics and visualization enables the application of PC, as well as FEM, not only in the pre - and post - process phase, but also in the interprocess phase. In such a case, the FEM becomes important as a real component of engineering design.

At the end of this part it is necessary to mention the works of O.C.Zienkiewicz [8], E.Hinton [9], P.G.Ciarlet, P.A. Raviart [10], A.Jennings [11], D.R.J. Owen, E.Hinton [12], J.Robinson [13], T.J.R.Hughes[14] and K.J.Bathe [15].

In the following period, several publications by authors all over the world [16-30] contributed to the solution of tasks in the FEM.

I myself have published more than 500 papers at scientific conferences and in journals in the field of the safety and reliability analyzes of structures under the extreme climatic loads, seismicity, terrorist attacks, fires and others considering the soil-fluid-structure interactions [31-38]. Since 1978 We have rich experience with teaching FEM at the Faculty of Civil Engineering STU in Bratislava. Initially, we used our own software packages in teaching of FEM, later we focused to professional packages (IDA NEXIS32, FEAT, RFEM, SAP2000, COSMOS, GT STRUDL and ANSYS).

At present, there are several world-famous programs in the field of FEM (ABAQUS, ADINA, ANSYS, ALGOR, CivilFEM, GT STRUDL, MARC, MSC PATRAN, NASTRAN, NISA II, COSMOS/M, SAP2000 and others), which contain the latest knowledge in computational analysis of static and dynamic problems, linear and nonlinear processes and bounded problems in mechanics. These programs belong to financially demanding programs and therefore in practice in Slovakia, limited application programs for structural practice are used (SCIA ESA PT, IDA NEXIS32, FEAT, RFEM and others...).

The applications listed here are illustrated under the ANSYS system [32-38], a product of SASI, in one of the largest and most widespread FEM systems in the world. ANSYS as one of the reference systems in the USA, EC countries and Japan is used for state and military projects. It is one of the strategic SW, the import of which is subject to licensing. On January 1, 1992, the congress committee in the USA gave its consent to the export of SW ANSYS to the then Czechoslovakia. Among other things, it is a certified system

with a quality certificate ISO 9001: 1994/7372 (US) also for nuclear technologies, and as a result its quality is controlled by the US NRC (US Nuclear Commission), which e.g. Prior to the launch of the new version, ANSYS prescribes approximately 3,500 system tests performed by independent companies. The great advantage of this system is that it relies on ongoing cooperation with a number of universities and research institutes in the USA, the EC and Japan.

In 1992, in a survey by the trade magazine "Machin Design, ANSYS received the highest award in the category of FE programs. For this reason, ANSYS became the mainstay of the FEM educational system NTC (National Technology Corporation) in the USA.

About the basic philosophy of the company. ANSYS, Inc. say aptly the words of its president "We don't sell software, we license technology".

One of the great advantages of this system is the possibility to use a database of results of experimental measurements of real physical properties of materials, not only depending on deformations, but also on time, temperature and stress levels.

The perspectives of the development of this system lie in the activities of the AUC (Ansys User Club), which groups individual users around top scientific and university workplaces and regularly organizes scientific conferences in the field of using FEM in the development of new promising technologies. The system is regularly expanded every year and supplemented with the latest knowledge in the field of structural and potential analyzes.

The development of the system is provided by the company. ANSYS, Inc. unified information system through the magazine "ANSYS news", "Infoplaner" for Europe and "FEM news to Use" for the Czech Republic and Slovakia.

ANSYS makes it possible to comprehensively solve problems in the field of strength calculations, analysis of temperature, acoustic, magnetic, electric, and piezoelectric fields and general potential problems. The system [17] contains more than 100 basic finite elements and P-elements (higher accuracies) in the plane and in space and contact elements with one-sided bonds and with friction on the contact surface. The ANSYS system is open and can communicate with various user programs and CAD systems via the pre- and post-processor. The system is compatible with NASTRAN, ADAMS, SIMPACK, ProENGINEER, ProFEA, PATRAN, AutoCAD and others.

The preprocessor allows to model the structure as a solid-shaped body (Solid modeling) using Boolean operations, general spline surfaces (Non-Uniform Rational B-splines), automatic meshing and adaptive networks.

The ANSYS system contains the basic elements in the element library, including pipe elements (for solving piping systems) and the following procedures:

- structural and dynamic analysis - static, modal and transient analysis of structures in combination with nonlinear calculations and stability problems (Nonlinear Buckling)
- nonlinear analysis - solution of material, geometric and

structural nonlinearity and one-sided contacts using Newton - Raphson, Adapt. Descent, Arc-length, Line Search methods

- thermal analysis - solution of stationary and non-stationary conduction, flow and radiation of heat in the field of linear and nonlinear Newton-Raphson method in combination with Crank-Nicholson-Euler theta integration method
- potential analysis - solution of electrostatic and magnetic fields and piezoelectric phenomena, problems of laminar and turbulent flow of gases, liquids and heat, seepage through porous materials
- optimization - solution of the optimal shape of the structure with a bound condition on stresses, deformations, or shape.

The ANSYS program has built-in adaptive network (adaptive meshing) and error estimation (patch test), resp. an automatic system for estimating the area discretization error in the energy standard of the whole structure, as well as estimating the error of voltage levels by individual elements. This technique makes it possible not only to indicate inaccuracies in discretization, singularity, and convergence problems, but also in automatic division correction, i. allows the optimization of the division of the area in terms of minimizing the error of the solution. It also gives the possibility of upper and lower estimation of the accuracy of the analyzed physical quantities. This methodology is a guarantee of regularity of performed analyzes.

From the point of view of teaching FEM at University, there is a basic question of what students should know in the process of creating a computational model and computational methods so that the result of the analyzes is acceptable in terms of accuracy and detail of analyzes.

Students, underestimating the theoretical base of FEM and computational methods, make many mistakes in practice in terms of creating a correct model, choosing computational methods, and interpreting the results.

And this is a fundamental problem that students must be knowledge before using professional FEM-based software.

Based on our pedagogical experience [8], confirmed by the opinions of colleagues from abroad, engineering students must know the theoretical foundations of FEM, as well as go through solving simple applications to understand the nature of problems in applying FEM to solve engineering problems.

In case of doctoral studies, it is necessary for students to be able to develop FEM by developing new element types, new material models, more accurate and efficient computational methods and complex solutions considering the interaction of various structures. This paper presents our experiences [8-10] from the teaching of the FEM for the doctoral students using ANSYS software.

II. THEORETICAL BASIS OF FEM

The finite element method (FEM) is one of the most effective variational methods for solving problems of continuum mechanics, as well as gases and liquids and other potential problems (electromagnetism, acoustics, heat, ...). Its

essence is the division of the structure, respectively. of a continuous body on a set of finite elements connected to each other in division nodes. Such a discrete system must satisfy the conditions of continuity and equilibrium in the division nodes.

The calculation process in the FEM can be divided into five phases:

1. Discretization of the structure to a finite number of elements
2. Approximation of deformation or force quantities on each element separately,
3. Integration of finite elements into the whole while maintaining the conditions of continuity of deformations,
4. Energy minimization - solution of conditional equations and determination of unknown nodal parameters,
5. Determination of unknowns by elements - calculation of internal forces on individual elements.

The finite element method is based on the variational principle of minimum potential energy or on the theorem on the virtual work of forces on displacements in the nodes of a discretized body. After discretization of the continuous body into finite elements (Fig.II.1), these must satisfy the conditions of continuity and equilibrium in the nodes of division.

Fig. 1 Discretization of the solid on finite elements

The total energy of the discretized system is obtained by summarizing the energies of the individual elements. The formulation of basic relations is based on the investigation of the virtual work of forces on the basic element two-dimensional.

Table 1. Approximation functions of 2D elements

2D elements	Aproximation
<p>Linear xy</p> <p>1</p> <p>pascal triangle</p>
<p>Quadratic xy</p> <p>1</p>

The total virtual work of forces on a given system is defined in the form

$$\delta \Pi = \int_V \{\delta \varepsilon\}^T \{\sigma\} dV - \int_V \{\delta u\}^T \{b\} dV - \int_S \{\delta u\}^T \{p\} dS, \quad (II.1)$$

where $\{\delta \varepsilon\}$ is the virtual strain vector, $\{\sigma\}$ is the stress vector, $\{\delta u\}$ is the virtual vector of the displacements, $\{b\}$ is vector of the volume forces, $\{p\}$ is the vector of the surface loads.

The vector of the displacements is approximate by the polynomial functions in the form

$$\{u\} = [\Phi] \{\alpha\}, \quad (II.2)$$

where $[\Phi]$ is polynomial matrix and $\{\alpha\}$ is vector of polynomial coefficient. The condition for the convergence of the approximate solution is that the approximation functions on the surface of the element are smooth (differentiable), continuous, completely, and symmetrical (independent of the orientation of the coordinate axes).

The complete polynomial of degree n can be defined in the following form for 1D, 2D and 3D elements :

- 1D elements

$$P_n(x) = \sum_{k=1}^{n+1} \alpha_k x^{k-1} \quad (II.3)$$

- 2D elements

$$P_n(x) = \sum_{k=1}^{(n+1)(n+2)} \alpha_k x^i y^j \quad \forall i, j = 0..n, i + j \leq n$$

- 3D elements

$$P_n(x) = \sum_{k=1}^{(n+1)(n+2)(n+3)} \alpha_k x^i y^j z^l \quad \forall i, j, l = 0..n, i + j + l \leq n$$

The vector of unknown displacement degrees-of-freedom $\{r\}$ is determined in the nodes of the element depending on the selected polynomial after substituting the specific coordinates of the nodes of the element into the matrix of the polynomial in (II.2)

$$\{r\} = [A] \{\alpha\} \Rightarrow \{\alpha\} = [A^{-1}] \{r\} \quad (II.4)$$

After substituting (II.4) into (II.2) we obtain the shape matrix of deformation parameters $[N]$ in the form

$$\{u\} = [\Phi] \{\alpha\} = [\Phi] [A^{-1}] \{r\} = [N] \{r\} \quad (II.5)$$

On the finite element, we generally define the displacement vector depending on the vector of initial displacements and displacements from the given load in the form

$$\{u\} = \{u_o\} + [N] \{r\}. \quad (II.6)$$

Then we express the strain vector $\{\varepsilon\}$ in the form

$$\begin{aligned} \{\varepsilon\} &= [\mathcal{D}]^T \{u\} - \{\varepsilon_o\} = \{\varepsilon_u\} - \{\varepsilon_o\} + [\mathcal{D}]^T [N] \{r\}, \\ \{\varepsilon\} &= \{\varepsilon_u\} - \{\varepsilon_o\} + [B] \{r\}. \end{aligned} \quad (II.7)$$

where $\{\varepsilon_u\}$ is the strain vector from the initial displacements, $[B]$ is the shape matrix of the strains. The initial strains $\{\varepsilon_o\}$ usually correspond to the influences of the body loads from temperature, creep, or shrinkage of the structural material.

The stress vector $\{\sigma\}$ is obtained from material equations $\{\sigma\} = \{\sigma_o\} + [D] \{\varepsilon\} = \{\sigma_o\} + [D] ([B] \{r\} + \{\varepsilon_u\} - \{\varepsilon_o\})$ (II.8) where $\{\sigma_o\}$ is the initial stress vector.

After substituting relations (II.5) to (II.8) into equations (II.1), we get the total virtual work of forces in the form

$$\begin{aligned} \delta \pi &= \{\delta r\}^T \left(\int_V [B]^T [D] [B] dV \{r\} + \int_V [B]^T \{\sigma_o\} dV + \right. \\ &+ \int_V [B]^T [D] \{\varepsilon_u\} dV - \int_V [B]^T [D] \{\varepsilon_o\} dV - \int_V [N]^T \{b\} dV - \\ &\left. - \int_S [\tilde{N}]^T \{p\} dS \right) \end{aligned} \quad (II.9)$$

On base of the virtual work theorem, the expression in relation (II.9) for an equilibrium force system acting on a body of volume V and surface S must be equal to zero. Then we have algebraic equations in form

$$\begin{aligned} \int_V [B]^T [D] [B] dV \{r\} + \int_V [B]^T \{\sigma_o\} dV + \int_V [B]^T [D] \{\varepsilon_u\} dV - \\ - \int_V [B]^T [D] \{\varepsilon_o\} dV - \int_V [N]^T \{b\} dV - \int_S [\tilde{N}]^T \{p\} dS = \{0\} \end{aligned} \quad (II.10)$$

where matrix $[\tilde{N}]$ is equal to matrix $[N]$ on the surface S only.

The relations (II.10) correspondent to the equilibrium forces in element nodes

$$[K] \{r\} = \{F\} = \{F_o\} + \{F_\sigma\} + \{F_u\} + \{F_\varepsilon\} + \{F_b\} + \{F_p\}, \quad (II.11)$$

where $[K]$ is the stiffness matrix of the element, $\{r\}$ is the vector of nodal deformation parameters of the element, $\{F\}$ is the vector of generalized forces in nodes, $\{F_o\}$ is the vector of singular forces in nodes. We have

$$\begin{aligned} [K] &= \int_V [B]^T [D] [B] dV, \quad \{F_\sigma\} = - \int_V [B]^T \{\sigma_o\} dV, \\ \{F_u\} &= - \int_V [B]^T [D] \{\varepsilon_u\} dV, \quad \{F_\varepsilon\} = \int_V [B]^T [D] \{\varepsilon_o\} dV, \\ \{F_b\} &= \int_V [N]^T \{b\} dV, \quad \{F_p\} = \int_S [\tilde{N}]^T \{p\} dS. \end{aligned} \quad (II.12)$$

The relations in (II.12) are defined in local coordinate system. It is necessary to transform the forces and the displacements from the local to the global coordinate system.

We define the vectors $\{r\}_{glob}$ and $\{F\}_{glob}$ in global coordinate system following

$$\{r\} = [T] \{r\}_{glob} \quad \text{and} \quad \{F\} = [T] \{F\}_{glob} \quad (II.13)$$

Substituting relations (II.13) into (II.11) we get

$$[K] [T] \{r\}_{glob} = [T] \{F\}_{glob} \quad (II.14)$$

and after multiplying the equations from the left by the matrix we have

$$[T]^T [K] [T] \{r\}_{glob} = [K]_{glob} \{r\}_{glob} = \{F\}_{glob} \quad (II.15)$$

where $[T]^T [T] = [1]$ in orthogonal coordinate system.

On the whole structure the total vector of deformation parameters in the nodes is defined. On base of the continuity of elements in the nodes of division we define the relationship between the vector $\{r^e\}_{glob}$ and $\{F^e\}_{glob}$ on the element "e" and the total vector $\{r_{tot}\}$ and $\{F_{tot}\}$ as follows

$$\{r^e\}_{glob} = [L^e] \{r_{tot}\} \text{ and } \{F^e\}_{glob} = [L^e] \{F_{tot}\}, \quad (II.16)$$

where the localization matrix $[L^e]$ is defined by values equal to 1 or 0 as follows

$$\{r^e\}_{glob} = [L^e] \{r_{tot}\} \text{ for } L_{mi}^e = \begin{cases} 1 & \forall r_m^e \equiv r_i^{tot} \\ 0 & \forall r_m^e \neq r_i^{tot} \end{cases} \quad (II.17)$$

If the structure consists of n -elements, then the total virtual work of forces is equal to the algebraic sum of virtual work of forces on individual elements

$$\delta\pi = \sum_{e=1}^n \delta\pi^e = \{\delta r_{tot}\}^T \left(\sum_{e=1}^n [L^e]^T [K^e]_{glob} [L^e] \{r_{tot}\} - \sum_{e=1}^n [L^e] \{F_{tot}\} \right) = 0 \quad (II.18)$$

From where we get the total condition equations of the balance of forces on a given structure

$$[K_{tot}] \{r_{tot}\} = \{F_{tot}\} \quad (II.19)$$

where the elements of the total stiffness matrix of the structure are obtained from the stiffness matrices of individual elements

$$K_{ij}^{tot} = \sum_{e=1}^n K_{km}^e L_{ki}^e L_{mj}^e \text{ for } L_{ki}^e = \begin{cases} 1 & \forall r_k^e \equiv r_i^{tot} \\ 0 & \forall r_k^e \neq r_i^{tot} \end{cases} \text{ and } L_{mj}^e = \begin{cases} 1 & \forall r_m^e \equiv r_j^{tot} \\ 0 & \forall r_m^e \neq r_j^{tot} \end{cases} \quad (II.20)$$

and elements of the total vector of generalized forces in the form

$$F_i^{tot} = \sum_{e=1}^n F_k^e L_{ki}^e \quad (II.21)$$

A. Solution of structural statics

In case of linear analysis of the structures with the elastic materials solution of the displacement degrees-of-freedom is based on a relationship (II.19) in form

$$\{r_{tot}\} = [K_{tot}]^{-1} \{F_{tot}\} \quad (II.22)$$

Next, the vector $\{r^e\}$ on element "e" is calculated from the relations (II.16) and (II.13). The vectors of strains and stress are calculated from the relations (II.7) and (II.8).

B. Solution of structural stability

The formulation of stability problem is simple and is based on the equilibrium condition on a deformable structure. In fact, it is a combination of the effect of the membrane (wall) and bending (plate) state of stress. We express the stiffness matrix of the element in the form $[[K] + [K_G]]$, where the matrix $[K_G]$ depends on the load vector $\{F\}$. When the load increases $\{F\} = \lambda \{F_o\}$, the stiffness matrix $[K_G]$ also changes proportionally. We express the problem of linear stability in the form of following equations

$$[[K] + \lambda [K_G]] \{r\} = \lambda \{F_o\} \quad (II.23)$$

In case of the collapse, the force vector does not act on the system and equation (II.23) changes to the shape

$$[[K] + \lambda [K_G]] \{r\} = \{0\} \quad (II.24)$$

The condition for the non-trivial solution of the equation (II.24) is that the determinant of the equilibrium condition matrix is equal to zero

$$\det[[K] + \lambda [K_G]] = 0 \quad (II.25)$$

The vector of critical forces is then expressed from the factor λ

$$\{F_{cr}\} = \lambda \{F_o\} \quad (II.26)$$

C. Solution of material nonlinearity

Material nonlinearity corresponds to the state when the dependence of the stress vector on the deformations is nonlinear. The vector of elastic strain for a nonlinear material is expressed in the form:

$$\{\varepsilon^{el}\} = \{\varepsilon\} - \{\varepsilon^{th}\} - \{\varepsilon^{pl}\} - \{\varepsilon^{cr}\} - \{\varepsilon^{sw}\}, \quad (II.27)$$

where $\{\varepsilon^{el}\}$ is elastic vector of strain, $\{\varepsilon\}$ - total vector of strain, $\{\varepsilon^{th}\}$ - vector of strain from temperature, $\{\varepsilon^{pl}\}$ - vector of plastic strain, $\{\varepsilon^{cr}\}$ - vector of strain from creep, $\{\varepsilon^{sw}\}$ - vector of strain from shrinkage.

The incremental plasticity theory defines physical equations for elasto-plastic material. These relationships are determined by boundary functions, plasticity functions and material reinforcement.

The stress increment vector is expressed depending on the incremental elastic strain vector in the form

$$\{d\sigma\} = [D_{el}] \{d\varepsilon^{el}\}, \quad (II.28)$$

where the elastic-plastic material matrix $[D_{el}]$ is defined in following form

$$[D_{ep}] = [D_e] - \frac{[D_e] \left\{ \frac{\partial Q}{\partial \sigma} \right\} \left\{ \frac{\partial F}{\partial \sigma} \right\}^T [D_e]}{-\frac{\partial F}{\partial \kappa} \sigma^T \left\{ \frac{\partial Q}{\partial \sigma} \right\} - C \left\{ \frac{\partial F}{\partial \alpha} \right\}^T \left\{ \frac{\partial Q}{\partial \sigma} \right\} + \left\{ \frac{\partial F}{\partial \sigma} \right\}^T [D_e] \left\{ \frac{\partial Q}{\partial \sigma} \right\}}$$

where Q is the plastic potential, F is the limited plastic function, κ is the strengthening factor (plastic work) and α is the change of the state of stress (location of the center of the limit function).

D. Solution of geometric nonlinearity

The geometric nonlinearities are nonlinearities in a structure or component due to a significant change in geometry from that proposed. The stiffness matrix $[K]$ is a function of the deformation vector $\{r\}$. The stiffness can also change due to the rotation of the structural elements or a change in the material properties of the elements.

The theory of large deformations is often used with the assumption of elasto-plastic behavior of structural elements. The displacement vector can be defined depending on the

vector of the original position $\{X\}$ of the element and the position after deformation $\{x\}$

$$\{u\} = \{x\} - \{X\} \quad (\text{II.29})$$

Using the incremental theory, the total strain vector is obtained from the strain vector in the previous step and the strain incremental vector

$$\{\varepsilon_n\} = \{\varepsilon_{n-1}\} + \{\Delta\varepsilon_n\}, \quad (\text{II.30})$$

where the strain vector increment is obtained for the shift at the midpoint, also between the two incremental steps n and $n-1$

$$\{\Delta\varepsilon_n\} = [B_{1/2}]\{\Delta r_n^u\}, \quad (\text{II.31})$$

where $\{\Delta r_n^u\}$ is the vector of the increment of displacements on the element corresponds to the incremental step and $[B_{1/2}]$ is the shape matrix of deformations for the mean values of the geometry approximation

$$\{X_{1/2}\} = \frac{1}{2}(\{X_n\} + \{X_{n-1}\}) \quad (\text{II.32})$$

This method is an excellent approximation of the logarithm of the deformation if the change in deformation is less than $\sim 10\%$. This method is often used for standard 2-D and 3-D spatial and shell elements.

III. PLANE STRESS

In two-dimensional problems, there are two special cases of stress and strain state - planar strain state ($\varepsilon_z = \gamma_{xz} = \gamma_{yz} = 0$) and planar stress state ($\sigma_z = \tau_{xz} = \tau_{yz} = 0$). For the planar state of stress or strain, the stress and strain vectors, which have affect to the deformation work are defined in the form

$$\{\sigma\} = \{\sigma_x, \sigma_y, \tau_{xy}\}^T \text{ and } \{\varepsilon\} = \{\varepsilon_x, \varepsilon_y, \gamma_{xy}\}^T \quad (\text{III.1})$$

The stress-strain relations are determined from 3D state considering the condition of the planar stress state or planar strain state

$$\{\sigma\} = [D]\{\varepsilon\} \text{ and } \{\varepsilon\} = [C]\{\sigma\} \quad (\text{III.2})$$

Table 3. Matrix of the physical relations

	Plane stress state
[C]	$\frac{1}{E} \begin{bmatrix} 1 & -\nu & 0 \\ -\nu & 1 & 0 \\ 0 & 0 & 2(1+\nu) \end{bmatrix}$
[D]	$\frac{E}{1-\nu^2} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & (1-\nu)/2 \end{bmatrix}$
	Plane strain state

[C]	$\frac{1}{E} \begin{bmatrix} 1-\nu^2 & -\nu(1+\nu) & 0 \\ -\nu(1+\nu) & 1-\nu^2 & 0 \\ 0 & 0 & 2(1+\nu) \end{bmatrix}$
[D]	$\frac{E}{(1-2\nu)(1+\nu)} \begin{bmatrix} 1-\nu & \nu & 0 \\ \nu & 1-\nu & 0 \\ 0 & 0 & (1-2\nu)/2 \end{bmatrix}$

Strain-stress relations for the plane stress and the plane strain state are obtained from relationships defined for 3D stress state. In case of the plane stress state the matrix [C] is similar with 3D stress state and in case of the plane strain state the matrix [D] is similar with 3D stress state (see tab.III.1). Then, for the strain-stress relations and the isotropic material we have following relation :

- Planar stress state

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \bar{E} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & (1-\nu)/2 \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} - \alpha_t \bar{E} T_{eff} \begin{Bmatrix} 1 \\ 1 \\ 0 \end{Bmatrix},$$

$$\bar{E} = E/(1-\nu^2), \quad (\text{III.3})$$

- Planar strain state

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \bar{E} \begin{bmatrix} 1-\nu & \nu & 0 \\ \nu & 1-\nu & 0 \\ 0 & 0 & (1-2\nu)/2 \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} - \alpha_t \bar{E} T_{eff} \begin{Bmatrix} 1 \\ 1 \\ 0 \end{Bmatrix},$$

$$\bar{E} = E/((1-2\nu)(1+\nu)) \quad (\text{III.4})$$

where E is Young modulus, ν - Poisson's ratio, α_t - thermal expansion coefficient, T_{eff} - effective temperature.

Fig. 2 Orientation of stresses on the surface with normal n

Normal and tangential stresses on the surface with normal n (Fig.III.1) can be expressed from the transformational relations

$$\sigma_n = (1/2)(\sigma_x + \sigma_y) + (1/2)(\sigma_x - \sigma_y) \cos 2\theta + \tau_{xy} \sin 2\theta,$$

$$\tau_n = -(1/2)(\sigma_x - \sigma_y) \sin 2\theta + \tau_{xy} \cos 2\theta. \quad (\text{III.5})$$

The orientation of the main stresses is obtained from the condition $\tau_n = 0$, and then

$$\tan 2\theta = 2\tau_{xy}/(\sigma_x - \sigma_y) \quad (\text{III.6})$$

The value of the principal stresses are calculated from the condition

$$\begin{vmatrix} \sigma_x - \sigma & \tau_{xy} \\ \tau_{xy} & \sigma_y - \sigma \end{vmatrix} = 0, \quad (III.7)$$

and then, we have

$$\sigma_{1,2} = \frac{\sigma_x + \sigma_y}{2} \pm \sqrt{\left(\frac{\sigma_x - \sigma_y}{2}\right)^2 + \tau_{xy}^2} \quad (III.7)$$

Stress and strain intensity (Saint Venant) is defined following

$$\sigma_I = \text{MAX}(|\sigma_1 - \sigma_2|, |\sigma_2|, |\sigma_1|),$$

$$\varepsilon_I = \text{MAX}(|\varepsilon_1 - \varepsilon_2|, |\varepsilon_2|, |\varepsilon_1|). \quad (III.8)$$

Effective stress and strain (Von Mises) is defined following

$$\sigma_{ef} = \left(\frac{1}{2} [(\sigma_1 - \sigma_2)^2 + (\sigma_2)^2 + (\sigma_1)^2] \right)^{\frac{1}{2}},$$

$$\varepsilon_{ef} = \frac{1}{1+\nu} \left(\frac{1}{2} [(\varepsilon_1 - \varepsilon_2)^2 + (\varepsilon_2)^2 + (\varepsilon_1)^2] \right)^{\frac{1}{2}}. \quad (III.9)$$

Fig. 3 The quadrilateral isoparametric 2D element.

The displacements and coordinates on the 2D isoparametric finite elements are defined as follows

$$u(\xi, \eta) = \sum_{i=1}^n N_i(\xi, \eta) u_i, \quad v(\xi, \eta) = \sum_{i=1}^n N_i(\xi, \eta) v_i \quad (III.10)$$

$$x(\xi, \eta) = \sum_{i=1}^n N_i(\xi, \eta) x_i, \quad y(\xi, \eta) = \sum_{i=1}^n N_i(\xi, \eta) y_i$$

where n is number of nodes on element. The shape functions for the quadrilateral linear element are

$$N_i(\xi, \eta) = 0.25(1 + \xi_i \xi)(1 + \eta_i \eta) \quad \forall i = 1..n \quad (III.11)$$

$$\sum_{k=1}^n N_k(\xi_i, \eta_i) = 1 \quad \text{for} \quad -1 \leq \xi_i \leq 1, -1 \leq \eta_i \leq 1.$$

The strain vector is calculated from (II.7) and (III.7)

$$\{\varepsilon\} = [\partial]^T \{u\} = [\partial]^T [N] \{r\} = [B] \{r\}. \quad (III.12)$$

The stiffness matrix of the quadrilateral element is obtained from (II.12), (III.2), (III.5) and (III.12)

$$[K] = \iint [B]^T [D] [B] dS = \int_{-1}^1 \int_{-1}^1 [B]^T [D] [B] |J| d\xi d\eta =$$

$$= \sum_{p=1}^{n_{\text{gauss}}} \sum_{q=1}^{n_{\text{gauss}}} W_p W_q [B(\xi_p, \eta_q)]^T [D] [B(\xi_p, \eta_q)] |J(\xi_p, \eta_q)| \quad (III.13)$$

where $[J]$ is Jacoby matrix, W_p, W_q are the weights in the Gauss nodes ξ_p, η_q of the numerical integration.

IV. PLATE BENDING

The plate bending theory is based on proposition of the plane stress/strain relations and depends on the plate type. We have

- thin plate $(1/50 < h/l < 1/10)$,
- semi-thick plate $(1/10 < h/l < 1/5)$,
- thick plate $(1/5 < h/l)$,

where h is the plate thickness and l is characteristic plate dimension. The solution of the thin plate is based on Kirchhoff's hypotheses and semi-thick plate on Reissner-Mindlin's hypotheses.

The displacements are defined on base of Kirchhoff in form

$$u(x, y, z) = z \theta_y(x, y), \quad v(x, y, z) = -z \theta_x(x, y), \quad (IV.1)$$

$$w(x, y, z) = w(x, y, 0).$$

where θ_x (or θ_y) is rotation of section about axis X (or Y).

The shear deformations in plane yz and zx are determined from the 3D state on base of Reissner-Mindlin's hypotheses

$$\gamma_{zx}(x, y, z) = \frac{\partial w(x, y, z)}{\partial x} + \frac{\partial u(x, y, z)}{\partial z}$$

$$\gamma_{zx}(x, y, z) = \frac{\partial w(x, y, 0)}{\partial x} + \theta_y(x, y, 0) = \gamma_{zx}(x, y, 0),$$

$$\gamma_{yz}(x, y, z) = \frac{\partial w(x, y, z)}{\partial y} + \frac{\partial v(x, y, z)}{\partial z}$$

$$\gamma_{yz}(x, y, z) = \frac{\partial w(x, y, 0)}{\partial y} - \theta_x(x, y, 0) = \gamma_{yz}(x, y, 0) \quad (IV.2)$$

Fig.. 4 Scheme of the plate displacement w and rotations θ_x , θ_y .

The rotation of section θ_x (or θ_y) is defined from (IV.2) by Mindlin's theory as follows

$$\theta_y(x, y) = -w_{,x}(x, y, 0) + \gamma_{zx}(x, y, 0) \quad (IV.3)$$

$$\theta_x(x, y) = w_{,y}(x, y, 0) + \gamma_{yz}(x, y, 0)$$

In case of Kirchhoff's hypotheses the shear deformations are neglected ($\gamma_{yz}(x, y, 0) = 0$ and $\gamma_{zx}(x, y, 0) = 0$) and we have

$$\theta_y(x, y) = -w_{,x}(x, y, 0) \quad \text{and} \quad \theta_x(x, y) = w_{,y}(x, y, 0) \quad (IV.4)$$

The strains are defined from 3D state in the following form

$$\varepsilon_x(x, y, z) = \frac{\partial u(x, y, z)}{\partial x} = z \frac{\partial \theta_y(x, y)}{\partial x} = z \theta_{y,x}, \quad (IV.5)$$

$$\varepsilon_y(x, y, z) = \frac{\partial v(x, y, z)}{\partial y} = -z \frac{\partial \theta_x(x, y)}{\partial y} = -z \theta_{x,y},$$

$$\gamma_{xy}(x, y, z) = \frac{\partial u(x, y, z)}{\partial y} + \frac{\partial v(x, y, z)}{\partial x} = z(\theta_{y,y} - \theta_{x,x}).$$

The stresses are defined for the plane stress state

$$\sigma_x(x, y, z) = \frac{E}{1-\nu^2}(\varepsilon_x + \nu\varepsilon_y) = \frac{Ez}{1-\nu^2}(\theta_{y,x} - \nu\theta_{x,y}), \quad (IV.6)$$

$$\sigma_y(x, y, z) = \frac{E}{1-\nu^2}(\varepsilon_y + \nu\varepsilon_x) = \frac{Ez}{1-\nu^2}(-\theta_{x,y} + \nu\theta_{y,x}),$$

$$\tau_{xy}(x, y, z) = \frac{E}{2(1+\nu)}\gamma_{xy} = \frac{E}{(1-\nu^2)}\frac{(1-\nu)}{2}z(\theta_{y,y} - \theta_{x,x}).$$

In case of the Mindlin's theory we have

$$\tau_{zx}(x, y, z) = \frac{E}{2(1+\nu)}\gamma_{zx} = G(w_{,x} + \theta_y), \quad (IV.7)$$

$$\tau_{zy}(x, y, z) = \frac{E}{2(1+\nu)}\gamma_{zy} = G(w_{,y} - \theta_x).$$

We define the internal forces on the plates as integral stress results in planes perpendicular to the median plane. The vector of internal forces contains 5 components:

- Intensity of bending moments

$$m_x(x, y) = \int_{-h/2}^{h/2} \sigma_x z dz = D_o(\theta_{y,x} - \nu\theta_{x,y}), \quad (IV.8)$$

$$m_y(x, y) = \int_{-h/2}^{h/2} \sigma_y z dz = D_o(-\theta_{x,y} + \nu\theta_{y,x}).$$

- Intensity of bending and torques moments

$$m_{xy}(x, y) = \int_{-h/2}^{h/2} \tau_{xy} z dz = D_o \frac{(1-\nu)}{2}(\theta_{y,y} - \nu\theta_{x,x}). \quad (IV.9)$$

- Intensity of shear forces

$$n_x(x, y) = \int_{-h/2}^{h/2} \tau_{xz} dz = Gh\beta(w_{,x} + \theta_y), \quad (IV.10)$$

$$n_y(x, y) = \int_{-h/2}^{h/2} \tau_{yz} dz = Gh\beta(w_{,y} - \theta_x),$$

where the plate stiffness $D_o = Eh^3 / (12(1-\nu^2))$ is defined in the forme and the shear coefficient β ($\beta = 5/6$) presents the effective shear surface of the section, i.e. the influence of the simplifying assumption on the constant behavior of shear stresses along the height of the plate cross-section on the deformation work of shear stress components.

Fig. 5 Scheme of the plate internal forces.

In the case of solving plates based on FEM, we approximate the components of the displacement vector on the finite element and we look for the solution of the state of stress and deformation of the plate in the form of algebraic equations after applying the virtual work theorem.

The displacement vector on the plate element we have

$$\{u\} = \{w, \theta_x, \theta_y\}, \quad (IV.11)$$

Next, the vector of the displacement degree-of-freedom on element is

$$\{r\} = \{w_1, \theta_{x1}, \theta_{y1}, \dots, w_n, \theta_{xn}, \theta_{yn}\}, \quad (IV.12)$$

where n - number of nodes.

The interpolation of the displacements on isoparametric finite elements is

$$\{u\} = \{w, \theta_x, \theta_y\}, \quad \theta_x = \sum \theta_{xi} N_i, \quad \theta_y = \sum \theta_{yi} N_i, \quad (IV.13)$$

where the shape functions N_i is the same as (III.11) for the quadrilateral linear element.

After integration in the z -direction, the 5 forces are defined

$$\{Q\} = \{m_x, m_y, m_{xy}, n_x, n_y\}^T \text{ and equivalent deformations}$$

$$\{q\} = \{\theta_{y,x}, -\theta_{x,y}, \theta_{y,y} - \theta_{x,x}, w_{,x} + \theta_y, w_{,y} - \theta_x\}^T, \text{ for which we have}$$

$$\{Q\} = [\tilde{D}] \{q\} \quad (IV.14)$$

$$[\tilde{D}] = D_o \begin{bmatrix} 1 & \nu & 0 & 0 & 0 \\ \nu & 1 & 0 & 0 & 0 \\ 0 & 0 & \frac{1-\nu}{2} & 0 & 0 \\ 0 & 0 & 0 & \frac{1-\nu}{2} \beta & 0 \\ 0 & 0 & 0 & 0 & \frac{1-\nu}{2} \beta \end{bmatrix}, \quad D_o = \frac{Eh^3}{12(1-\nu^2)}.$$

The stiffness matrix of the quadrilateral plate element is obtained from (II.12), (III.5), (III.6) and (III.7)

$$(IV.15)$$

$$[K] = \int \int [B]^T [D] [B] dS = \int \int_{-1-1}^{11} [B]^T [D] [B] |J| d\xi d\eta =$$

$$= \sum_{p=1}^{ngaus} \sum_{q=1}^{ngaus} W_p W_q [B(\xi_p, \eta_q)]^T [D] [B(\xi_p, \eta_q)] |J(\xi_p, \eta_q)|$$

where $[J]$ is Jacoby matrix, W_p, W_q are the weights in the Gauss nodes ξ_p, η_q of the numerical integration.

V. PROGRAMABLE LANGUAGE IN ANSYS SOFTWARE

Solving the structures in ANSYS can be accelerated and streamlining the use of macros. Macro is a text file with a MAC extension. In macro, the ANSYS commands are described using special language, which is equivalent to the FORTRAN commands. However, in some cases we do not only fill the ANSYS orders, but it is necessary to manage or

directly acquire information on the structure model and to decide on the next procedure of calculation. APDL (ANSYS Parametric Design Language) is a means that makes it possible to address the above-described problems. It contains assignment, cycle, branches, read and write parameter, file writing, and so on. The most used commands are described in following applications.

VI. APPLICATIONS

Make an analysis of the state of the stress/strain I-beam on the FEM based using shell elements (SHELL181). Consider with steel material properties with S235 ($E = 210\text{GPa}$, $f_y = 235\text{MPa}$, $\rho = 7850\text{kgm}^{-3}$) and following geometric dimensions of the I-beam: $h = 0.5\text{m}$ - wall height, $b = 0.2\text{m}$ - width, $t_s = 0.01\text{m}$ and $t_p = 0.016\text{m}$ - thickness wall, $k = 8$ -ratio.

Solve the following tasks:

- Modeling in FEM – prepare FEM model of I-beam.
- Static analysis - solve the maximum load value from the maximum beam deflection ($u_{\text{lim}} = l / 250$).
- Stability analysis - solve the critical load value for defined loads.
- Nonlinear analysis - solve the maximum loads considering material and geometric nonlinearity.

Fig. 6 The schema of I-beam

A. Ansys procedures

Jobname: IBEAM

File : MBEAM.MAC

```
*****
!*
!*** FEM in ANSYS - Modeling of I-Beam ***
!*
*****
!
! Calculation procedures: 1. Preprocessor (/PREP7)
!                        2. Solution (/SOLU)
!                        3. Postprocessor (/POST1)
!
!
! Units : kN, m, kPa
! Element Type : SHELL181
! Input Data:
! - Material : EX_, MI_, Dens_
! - Stress limit : Fay_, Fau_
```

```
! - Dimensions : L_, B_, h_, ts_, tp_
! - Loads : Fy_, p_
! - Loads cases : Lc1_, Lc2_, Lc3_, Lc4_, Lc5
!
!*****
! ** NOTE:
!* 1. Software ANSYS does not distinguish small
!* and uppercase letters
!* 2. Text after “!” is description not command
!* 3. Parameter in form %XXX% is interpreted by the
!* content of parameter XXX
!*****
!
! FINISH
/CLEAR
/TITLE,1. FEM model of I-beam
/TRIAD, RBOT ! To set up a global coordinate system
!
!*** Definition of auxiliary file “Job.tit”
!
! Define file name Job=IBeam
*GET, Job, ACTIVE, 0, JOBNAM, , START
! Creating of file “IBeam.tit” with following text
*CREATE, %Job%, tit
/TITLE, 1. Solution of the wall I-Beam *
*END ! Finish editation of file “IBeam.tit”
!
! Creating of file “IBeam.txx” with following text
*CREATE, %Job%, txx
/TITLE, 1. Solution of the wall I-Beam * Lc%Lc_% *
*END ! Finish editation of file “IBeam.txx”
!
Ex_ = 210E6 ! Yung modulus [kPa]
Mi_ = 0.2 ! Poisson ratio
Dens_ = 7.85 ! Density [t/m^3]
ag_ = 9.81 ! Gravitation [m/s^2]
Alpx_ = 1.2E-5 ! Coefficient of thermal expansion [1/deg]
Fay_ = 235000. ! Yield strength of steel
Fau_ = 360000. ! Strength limit of steel
Gm1_ = 1.1 ! Reliability factors
Gm2_ = 1.3
Rs_min = Fay_ / Gm1_ ! Design value of strength in pressure
Rs_max = Fau_ / Gm2_ ! Design value of strength in tension
!.....
k_ = 4 ! Ratio L/h
h_ = 0.4 ! Wall with [m]
L_ = k_ * h_ ! Beam span [m]
B_ = h_ / 2 ! Flange dimension [m]
ts_ = h_ / 20 ! Wall thickness [m]
tp_ = 2 * ts_ ! Flang thickness [m]
x1_ = h_ / 3 ! 1. Support coordinate
x2_ = L_ - h_ / 3 ! 2. Support coordinate
De_ = h_ / 8 ! Element dimension
!.....
Ru_max = L_ / 250 ! Deflection limit
!.....
Fy_ = -1 ! Singular force [kN]
```

```
p_1      ! Surface loads [kN/m]
!.....
N_lc=3    ! Load Number of load cases
Lc1_ =1   ! 1. Load case - self weight
Lc2_ =2   ! 2. Load case - singular force Fy
Lc3_ =3   ! 3. Load case - surface load p
Lc4_ =4   ! 4. Load combination - LC1_+LC2_
Lc5_ =5   ! 5. Load combination - LC1_+LC3_
M_lc=Lc5_ ! Total number of load cases
!.....
*DIM,LP_,M_lc ! Vector dimension - LP
LP_(1)=1,2,3,4,5 ! Vector values
!.....
! M_Lc=N_lc+2
*DIM,U_max,,M_lc ! Vector dimension - U_max
*DIM,S_max,,M_lc ! Vector dimension - S_max
*DIM,S_min,,M_lc ! Vector dimension - S_min
*DIM,Nu_max,,M_lc ! Vector dimension - Nu_max
*DIM,Ns_max,,M_lc ! Vector dimension - Ns_max
*DIM,Ns_min,,M_lc ! Vector dimension - Ns_min
*DIM,RFu_,M_lc ! Vector dimension - RFu_max
*DIM,RFt_,M_lc ! Vector dimension - RFt_max
*DIM,RFc_,M_lc ! Vector dimension - RFc_min
!.....
! _____ PREPROCESSOR
! Main Menu>Preprocessor>
FINISH
/PREP7      ! Start of Preprocessor
!.....
ET,1,SHELL181 ! Define element type 1.
ETLIST      ! Listing all element types
!
SECTYPE,1,SHELL ! Define section type 1.
SECDATA,ts_ ! Wall thickness - ts_
SECTYPE,2,SHELL
SECDATA,tp_ ! Flange thickness - tp_

MP,EX,1,Ex_ ! Material properties 1. - steel
MP,DENS,1,Dens_ ! Density
MP,NUXY,1,Mi_ ! Poisson ratio
MPLIST      ! Listing of material properties
!.....
! _____ SOLID MODELING

RECTNG,0,L_0,h_ ! Define rectangular area 1.
AATT,1,1,1, ! Add all attribute to area 1.
ASELL,NONE ! Unselect all areas
WPLANE,,,,,1,,,,-1 ! Define work plane ZX in GSS
WPOFFS,-B_/2 ! Move wplane for WY=-B_/2
RECTNG,0,L_0,B_ ! Define rectangular area 2.
WPCSYS ! Define work plane in GSS
AGEN,2,ALL,,h_ ! Copy area of lower flang to upper
ALLSEL ! Select all areas
NUMMRG,KP ! Bonding all keypoints with same
! coordinates
APTN,all ! Creating intersection of all areas
ASEL,U,REAL,,1 ! Unselect all areas for REAL=1
AATT,1,2,1, ! Add all attribute to selected areas
```

```
ALLSEL
/NUMBER      ! Define viewing of entite numbers
/PNUM,LINE,1 ! Activation viewing of line number
LPLOT        ! Plot of line in monitor

!*** Transformation of solid model to FEM ***

LESIZE,ALL,De_ ! Define discretisation of lines
LPLOT
AMESH,ALL ! Discretisation of ares to FE
ELIST      ! Listing elements
EPLOT      ! Plot elements
/NUMBER,1 ! Define viewing of entites in color
/PNUM,REAL,1 ! Activation viewing lines in color
SAVE ! Write database to IBEAM.db in binar
CDWRITE ! Write database to IBEAM.cdb in ascii
FINISH ! Finish of PREPROCESSOR
!
! _____
B. Solution of structural statics
Jobname: IBEAM
File : SBEAM.MAC
!*****
!*
!*** FEM in ANSYS - Static SOLUTION ***
!*
!*****
RESUME ! Read of I-Beam model from IBEAM.DB
! _____ SOLUTION
!
/SOLU ! Start of Solution procesing
LSCLEAR,ALL ! Clear all load cases
KP_=KP(L_,0.,0.) ! Define of keypoint (KP) number for
! coordinates (x,y,z)
KSEL,S,KP,,%KP(0.,0.,0.)% ! Select keypoint in coordinate
KSEL,A,KP,,KP_ ! Add 1. keypoint KP_
DK,all,UX,,,UY,UZ ! Define boundary in these active KP
LSEL,S,LOC,x,0 ! Select lines for boundary definition
LSEL,A,LOC,x,L_ ! Add selection lines for x=L_
LSEL,R,LOC,y,0 ! Reselect lines for y=0
LPLOT ! Plot selected lines
DL,ALL,,UY ! Define boundary for selected lines
!
NSEL,S,LOC,Y ! Reselect nodes for Y=0
NSEL,U,LOC,X,x1_,x2_ ! Unselect nodes for x1_<X<x2_
NPLOT ! Plot selected nodes
D,ALL,UY ! Define boundery on selected nodes
ALLSEL
SAVE ! Write database to IBEAM.db in binar
CDWRITE ! Write database to IBEAM.cdb in ascii
! _____ LOAD CASE* Lc1
!
LC_=LC1_ ! 1. Load case - self weight
/INP,,txx ! Read of title file IBEAM.txx
ACEL,,ag_ ! Define of gravity acceleration
LSWRITE,Lc_ ! Write load case to IBEAM.S01
ACEL ! Deleting of gravity acceleration
! _____ LOAD CASE* Lc2
```

```

!
LC_=LC2_      ! 2.Load case – Force Fy
/INP,,txx
NSEL,S,NODE,,%NODE(L_/2,h_,0.)%  ! Selecting of node
!                               for coordinate (x,y,z)
F,ALL,FY,Fy_   ! Define of load by force Fy
ALLSEL
LSWRITE,Lc_    ! Write loads to file IBEAM.S02
FDELETE,ALL,ALL ! Delete all forces
!_____LOAD CASE * Lc3
!
LC_=LC3_      ! 3.Load case by pressure p
/INP,,txx
ASEL,S,LOC,Y,H_ ! Selection area for Y=h
APLOT         ! Plot of selected areas
SFA,ALL,2,PRES,p_ ! Definition of pressure p
ALLSEL
SFTRAN       ! Transformation of load from areas to FE
/PSF,PRES,,2 ! Definition of viewing of press by darts
EPLOT
LSWRITE,Lc_  ! Write load case to IBEAM.s02
SFADDELE,ALL,2,ALL ! Deliting of all surface loads
!_____SOLVE
!
LSSOLVE,Lc1_,Lc3_ ! Solution load cases Lc1-Lc3
FINISH           ! The results are writing in
IBEAM.rst
!
!*** Design check (Elastic state)
!
! Creating macrum to writing output data to file "LC_".prn
!*GO,:end       ! GOTO to “:end”
!
!***** Creating of Macra : Post1.mac
*CREATE,Post1,mac
!_____
!_____
POSTPROCESOR
! Main Menu>Postprocesor>
!
/POST1          ! Start of Postprocesor
ALLSEL
!
LCDEF,Lc1_,1,1  ! Define of load cases in IBEAM.rst
LCDEF,Lc2_,2,1
LCDEF,Lc3_,3,1

*DO,i_,1,N_Lc  ! Cyklus for i_=1 to N_Lc
!
Lc_=Lc%i_%,_  ! Define value of parameter Lc_
LCASE,Lc_      ! Read results for Lc_
/INP,,txx
LCWRITE,Lc_    ! Write output to IBEAM.C0X
!              (X=1,2,...,N_Lc)
SHELL,MID      ! Define of output for midle level
NSORT,U,Y,,1   ! Reorganisation of results by absolute
!              value of UY displacement

*GET,U_max(Lc_),SORT,,MAX ! Parameter U_max = max.
!              displacement
*GET,Nu_max(Lc_),SORT,,IMAX ! Read node number with
!              max. displacement
NSORT,S,1,     ! Reorganisation of results by principal
!              stress values S1 (max)
*GET,S_max(Lc_),SORT,,MAX ! Read max. principal stress
!              S_max
*GET,Ns_max(Lc_),SORT,,IMAX ! Read node number with !
the max stress
NSORT,S,3,     ! Reorganisation of results by principal
!              stress values S3 (min)
*GET,S_min(Lc_),SORT,,MIN ! Read min. principal stress
!              S_min
*GET,Ns_min(Lc_),SORT,,IMIN ! Read node number with
!              the min stress S3
! Reliability test for max. displacement and stresses
RFu_(Lc_)=abs(U_max(Lc_))/Ru_max
*IF,S_max(Lc_),GT,0.,THEN
  RFt_(Lc_)=S_max(Lc_)/Rs_max
*ENDIF
*IF,S_min(Lc_),LT,0.,THEN
  RFc_(Lc_)=abs(S_min(Lc_))/Rs_min
*ENDIF
!
/OUTPUT,%Job%_%Lc_%,prn ! Write the output data to file
!              IBEAM_%Lc_%.prn
PRNSOL,U        ! Print results of displacement
PRRSOL          ! Print result forces
PRNSOL,S        ! Print stresses
PRNSOL,S,PRIN   ! Print of principal stresses
/OUTPUT         ! Redirection of output to display
!
*ENDDO          ! Cyclus finish for i_=N_Lc
!_____
!_____DESIGN RECAPITULATION
!_____Design of LC4_ and LC5_
!_____
*DO,i_,N_Lc+1,M_lc ! Cyclus pre i_=N_Lc+1 to M_Lc
!
LCASE,Lc1_       ! Read results for LC1_
Lci_=i_-N_lc+Lc1_
Lc_=i_
!
! Read stress from LC1_ in the node with max. stress for Lci_
*GET,S_max(Lc1_),NODE,%Ns_max(Lci_)%S,1
! Read stress from LC1_ in the node with min. stress for Lci_
*GET,S_min(Lc1_),NODE,%Ns_min(Lci_)%S,3
! Read disp. from LC1_ in the node with max. stress for Lci_
U_max(Lc1_)=UY(Nu_max(Lci_))
!
!*** Calculation od reliability factors for limit states
!
*IF,U_max(Lc1_)*U_max(Lci_),GT,0.,THEN
  RFu_(Lc_)=abs(U_max(Lci_))/(Ru_max-abs(U_max(Lc1_)))
*ELSE
  RFu_(Lc_)=abs(U_max(Lci_))/(Ru_max+abs(U_max(Lc1_)))
*ENDIF

```

```

*IF,S_max(Lci_),GT,0.,THEN
  RFt_(Lc_)=abs(S_max(Lci_))/(Rs_max-S_max(Lc1_))
*ENDIF

*IF,S_min(Lci_),LT,0.,THEN
  RFc_(Lc_)=abs(S_min(Lci_))/(Rs_min-abs(S_min(Lc1_)))
*ENDIF

  RF_=RFu_(Lc_)
*IF,RF_,LT,RFt_(Lc_),THEN
  RF_=RFt_(Lc_)
*ENDIF

*IF,RF_,LT,RFc_(Lc_),THEN
  RF_=RFc_(Lc_)
*ENDIF

SF_=1.
*IF,RF_,NE,0.0,THEN
  SF_=1./RF_
*ENDIF
!
! Multiplication of Lci_ load by parameter SF_
LCFACT,Lci_,SF_
! Calculate of limit loads (LC_=LC1_+SF_*LCi_)
LCOPER,ADD,Lci_
LCWRITE,LC_ ! Write load to the file IBEAM.C0X
!
!*** Check of calculation of extreme values LC4_ and LC5_
!
SHELL,MID ! Define calculation output for midle level
NSORT,U,Y,,1 ! Sort nodes for abs. value of disp. UY
*GET,U_max(Lc_),SORT,,MAX ! Set "U_max"
! Set node number with max. disp U_max
*GET,Nu_max(Lc_),SORT,,IMAX
NSORT,S,1 ! Sort nodes for max. stress S1
! Set max. stress value S_max
*GET,S_max(Lc_),SORT,,MAX
! Set node number with max. stress S_max
*GET,Ns_max(Lc_),SORT,,IMAX
!
NSORT,S,3 ! Sort nodes for min. stress S3
! Set min. stress value S_min
*GET,S_min(Lc_),SORT,,MIN
! Set node number with max. stress S_min
*GET,Ns_min(Lc_),SORT,,IMIN
!
! Calculation of the ratio extrem values/limit values of stress
! and displacement
!
  RFu_(Lc_)=abs(U_max(Lc_))/Ru_max
*IF,S_max(Lc_),GT,0.,THEN
  RFt_(Lc_)=S_max(Lc_)/Rs_max
*ENDIF
*IF,S_min(Lc_),LT,0.,THEN
  RFc_(Lc_)=abs(S_min(Lc_))/Rs_min
*ENDIF
!.....
! Define the output file IBEAM_"Lc_"_.prn
/OUTPUT,%Job%_%"Lc_"_.prn
PRNSOL,U
PRRSOL
PRNSOL,S
PRNSOL,S,PRIN
/OUTPUT ! Redirect output to the screen

*ENDDO ! Finish of the cycle for i_=M_Lc
!.....
!
! _____ Design recapitulation
!
!*** Write the design recapitulation to the file IBEAM.prn
!
/OUTPUT,%Job%,prn ! Define the poutput file
!
  *VWRITE,'Design c','heck of ','the I-Be','am relia','bility
u','nder max','imum sta','tic load','s'
(30X,9A8)
  *VWRITE,' '
(1X,2A8)
  *VWRITE,'Ru_lim =',Ru_max,'Rc_lim =',Rs_min,'Rt_lim
=',Rs_max
(22X,A8,E12.5,', ' ,15X,A8,E12.5,', ' ,15X,A8,E12.5)
  *VWRITE,' '
(1X,2A8)
  *VWRITE,'LC',LP_(1),' Node =',Nu_max(1),' U_max
=',U_max(1),' Node =',Ns_min(1),' S_min =',S_min(1),' Node
=',Ns_max(1),' S_max =',S_max(1),
(1X,A2,F2.0,' ' ,A8,F5.0,' ' ,A8,E12.5,', ' ,A8,F5.0,'
',A8,E12.5,', ' ,A8,F5.0,' ' ,A8,E12.5)
  *VWRITE,' '
(1X,2A8)
  *VWRITE,'LC',LP_(1),' Node =',Nu_max(1),'Ru_max
=',RFu_(1),' Node =',Ns_min(1),'Rc_min =',RFc_(1),' Node
=',Ns_max(1),'Rt_max =',RFt_(1),
(1X,A2,F2.0,' ' ,A8,F5.0,' ' ,A8,E12.5,', ' ,A8,F5.0,'
',A8,E12.5,', ' ,A8,F5.0,' ' ,A8,E12.5)
!
/OUTPUT ! Redirect output to the screen
/SHOW,TERM ! Redirect graphic output to the screen
/SHOW,PNG ! Redirect graphic output to IBEAMxx.png
/TRIAD,RBOT ! Define GSS on right top level of the screen
!/CMAP,INVERT,CMAP ! Define blanc background
/RGB,INDEX,100,100,100,0 ! Define blanc background
/RGB,INDEX,0,0,0,15 ! Change color number 0 - 15
/AUTO
/INP,,tit ! Read text from the file IBEAM.tit
!
EPLOT ! Plot elements
!
!.....
*DO,i_,1,M_lc ! Start of cycle for i_ = 1 to M_lc
  Lc_=Lc%/i_ %_ ! Set load case Lc_
  LCFILE,Lc_ ! Set file with ouput data for Lc_
  LCASE,Lc_ ! Read outup data for Lc_

```

```

/INP,,txx      ! Read title name for Lc_
SHELL,MID      ! Define output data for shell middle level
!
PLDISP,2       ! Plot deformation shape of IBeam
!
PLNSOL,S,1     ! Plot principal stress S1 - max
PLNSOL,S,3     ! Plot principal stress S1 - min
PLNSOL,S,eqv   ! Plot principal stress Se - Mises
*ENDDO         ! Finish cyclus for i_ = M_lc
!
/SHOW,TERM     ! Redirect graphic output to the screen
!/CMAP,ORIG,CMAP ! Set black background – original
/INP,,tit      ! Read text from the file IBEAM.tit
EPLOT          ! Plot elements
!
FINISH         ! Finish Posptocesor
!
/RGB,INDEX,100,100,100,15 ! Set original color number
/RGB,INDEX,0,0,0,0 ! Change color number 0 and 15
*END           ! Finish editor mode
!
!***** Finish Macro Postl.mac
!
:END           ! Adress skip from point *GO :END
Postl         ! Run of macro Postl.mac
!
!_____Graphic output
!_____FINIS
H
!/EXIT,NOSAV   ! Close ANSYS without write database
!/EXIT        ! Close ANSYS with writing database
!

```

Recapitulation of results :

Fig. 7 FE Model of I-Beam

Fig. 8 Deformation shape of I-Beam under impact of Fy

C. Solution of structural stability

Jobname: VBEAM

File : VBEAM.MAC

```

!*****
!*
!*** FEM in ANSYS - Stability SOLUTION ***

```

```

!*
!*****
RESUME IBEAM.DB ! Read of I-Beam model
!
!_____SOLUTION
!
!*** Definition of auxiliary file "Job.tit"
!
! Define file name Job=VBEAM
*GET,Job,ACTIVE,0,JOBNAM,,START
! Creating of file "VBEAM.tit" with following text
*CREATE,%Job%,tit
/TITLE,2. Solution stability of the wall I-Beam *
*END ! Finish editation of file "VBEAM.tit"
!
! Creating of file "VBEAM.txx" with following text
*CREATE,%Job%,txx
/TITLE,2. Solution stability of the wall I-Beam * Lc%Lc_% *
*END ! Finish editation of file "VBEAM.txx"
!
!_____SOLUTION
! Main Menu>Solution>
/SOLU          ! Start solution
ANTYPE,STATIC ! Define Static analysis
PSTRES,ON     ! Calculation of static matrix
ALLSEL
SAVE          ! Save model to file VBEAM.DB
CDWRITE       ! Write model to file VBEAM.CDB
!_____Lod case
!
!*** Definition of load case with force Fy and self weight
!
Lc_=1
ACEL,,ag_     ! Define gravitation acceleraqtion
! Select node at middle beam and top level for force Fy
NSEL,S,NODE,,%NODE(L_/2,h_,0.)%
F,ALL,FY,Fy_ ! Define 1. Load case with force Fy
ALLSEL
LSWRITE,Lc_   ! Write load case to VBEAM.S01
!_____Solve stability for LC1
LSREAD,Lc_
SOLVE,Lc_     ! Run prestressed calculation
FINISH
!_____POSTPROCESSOR
/POST1        ! Start postprocesor
ALLSEL
SET,1         ! Set load case Lc_
LCWRITE,Lc_   ! Write Load case to VBEAM.C01
FINISH
!_____SOLVE CRITICAL LOADS
!
/SOLU
ANTYPE,BUCKLE ! Linear analysis
!*** Calculate 1.eigenvalue using Lanczos method
BUCOPT,LANB,1
MXPAND,1      ! Expand 1. mod
SOLVE
!_____

```

```
!
!*** OUTPUT DATA TO VBEAM.prn
!
Fcre_=1          ! Exact solution of critical loads
*GET,Fcr_,MODE,1,FREQ
*STATUS,PARM
*DIM,LABEL,CHAR,1,2
*DIM,Value,,1,3
LABEL(1,1)='Fcr '
LABEL(1,2)='kN '
*VFILL,Value(1,1),DATA,Fcre_
*VFILL,Value(1,2),DATA,Fcr_
*VFILL,Value(1,3),DATA,ABS(Fcr_/Fcre_)
FINISH          ! Finish buckling solution
!
!*GO,:END      ! Goto :end if macro Postlv.mac exist
!              ! Creating of the macro Postlv.mac
!
*CREATE,POSTLV,MAC ! Write following commands to
!              ! the file "Postlv.mac"
FINISH
/POST1          ! Open postprocessor
!
/OUTPUT,%Job%,prn ! Write output to file Vbeam.prn
/COM,
/COM,
!----- Kritikal load of wall I-Beam -----
/COM,
/COM,          | Exakt | ANSYS | Pomer
/COM,
*VWRITE,Label(1,1),Label(1,2),Value(1,1),Value(1,2),Value
(1,3)
(1X,A8,A8,' ',F10.3,' ',F10.3,' ',1F5.3)
/COM,-----
/OUTPUT        ! Redirect output to terminal
!
/SHOW,TERM     ! Redirect output to terminal
/SHOW,PNG      ! Redirect output to the file VBEAMXX.png
/TRIAD,RBOT    ! Show GSS at right botom figure
/RGB,INDEX,100,100,100,0 ! Set the black background
/RGB,INDEX,0,0,0,15      ! Change code of color 15
/AUTO
/INP,,tit      ! Read the title from VBEAM.TIT
!
ALLSEL
EPLOT          ! Plot the elements
!
LCASE,Lc_      ! Read data from file VBEAM.LC01
SHELL,MID      ! Define results for shell midle
PLDISP,2       ! Plot original model and after deformation
PLNSOL,S,1     ! Plot principal stress S1
PLNSOL,S,3     ! Plot principal stress S3
PLNSOL,S,EQV   ! Plot equivalent stress
/SHOW,TERM     ! Set graphic output to terminal
FINISH         ! Finish Postprocessor
!
!-----
```

```
! Changing color set to original
/RGB,INDEX,100,100,100,15
/RGB,INDEX,0,0,0,0
*END
!***** Finish of Macro
!
:END          ! Goto this line if *GO,:END
!
Postlv       ! Run macro Postlv.mac
!----- FINIS
H
/EXIT,NOSAV  ! Exit task from ANSYS
```

Recapitulation of results :

Fig. 9 Equivalent stress of I-Beam under impact of Fy

D. Solution of material and geometric nonlinearity

```
Jobname: NBEAM
File      : NBEAM.MAC
!*****
!*
!*** FEM in ANSYS - Nonlinearity SOLUTION ***
!*
!*****
RESUME IBEAM,DB    ! Read of I-Beam model
!----- SOLUTION
!
! Load factor defined in input parameters of macro
kf_=1.
*IF,ARG1,GT,0,THEN
kf_=ARG1
*ENDIF
!
!*** Definition of auxiliary file "Job.tit"
!
! Define file name Job=NBEAM
*GET,Job,ACTIVE,0,JOBNAM,,START
! Creating of file "NBEAM.tit" with following text
*CREATE,%Job%,tit
/TITLE,3. Nonlinear solution of the wall I-Beam *
*END          ! Finish editation of file "NBEAM.tit"
!
E01_=Fay_/Ex_ ! Strain at level Fay_
E02_=0.01     ! 1.Strain at level Yield strength
E03_=0.02     ! 2.Strain level at strength limit
!
TB,MISO,1,1,3 ! Define material diagram for MISO
! TBTEMP,20,      ! Define temperature
```

```

TBPT,DEFL,E01_,Fay_ ! TBPT,DEFL,E01_,Fay_
TBPT,DEFL,E02_,Fau_ ! TBPT,DEFL,E02_,Fau_
TBPT,DEFL,E03_,Fau_
TBPLOT,MISO,1
!
!*** Plastic section modul
Wpy=B_*tp_*(h_+tp_)+ts_*(h_**2)/4
! *** Singular force in middle beam simple formulation
Fy_=-4*Wpy_*Rs_max/L_
Fy_=Fy_*kF_ ! Modification for the load convergence
!
SAVE ! Save the database to NBEAM.DB
CDWRITE ! Write the ascii model to NBEAM.CDB
! _____NONLINEAR SOLUTION
!
ANTYPE,STAT,NEW
NLGEOM,ON ! Nonlinear geometric solution
NROPT,FULL,,ON ! N-Raphson full solution method
!*** Define increment and iteration for "AUTOTS,OFF"
SOLCONTROL,ON
NSUBST,10,100,1 ! Define max. number of step (10),
max.iteration (100) and number step to write results (1)
! NSUBST,100,1000,10 ! Alternative solution
! OUTRES,ALL,LAST ! Writing results of last iteration
and each step
OUTRES,ALL,ALL ! Alternative
AUTOTS,ON ! Automatic steping system is on
! Write all parameters to file NBEAM.LS01
LSWRITE,LC_
!
SOLVE ! Run solution
FINISH ! Finish solutkion
!.....
! Write new macro "Post1n.mac"
*CREATE,POST1N,MAC
!.....
! _____POSTPROCESOR
! Main Menu>Postprocesor>
!
/POST1 ! Start postprocesor mode
ALLSEL
!
Lc_=1 ! Number of load case
SET,1,LAST ! Read results data for last iteration
NSORT,U,Y,,1 ! Sort displacement abs(Uy) in order
! * Read max. displacement value into a vector U_max
*GET,U_max(Lc_),SORT,,MAX
! * Read node number with max. dis. into a vector Nu_max
*GET,Nu_max(Lc_),SORT,,IMAX
NSORT,S,1, ! Sort principal stress S1 in order
! * Read max. stress into a vector S_max
*GET,S_max(Lc_),SORT,,MAX
! * Read node number into a vector Ns_max
*GET,Ns_max(Lc_),SORT,,IMAX
NSORT,S,3, ! Sort principal stress S3 in order
! * Read min. stress into a vector S_min
*GET,S_min(Lc_),SORT,,MIN
! * Read node number into a vector Ns_min

*GET,Ns_min(Lc_),SORT,,IMIN
!* Calculation of reliability factors
RFu_(Lc_)=abs(U_max(Lc_))/Ru_max
*IF,S_max(Lc_),GT,0.,then
Rft_(Lc_)=S_max(Lc_)/Rs_max
*ENDIF
*IF,S_min(Lc_),LT,0.,then
RFc_(Lc_)=abs(S_min(Lc_))/Rs_min
*ENDIF
! Output redirection into file "NBEAM_LC_1.PRN"
/OUTPUT,%Job%_%Lc_%.prn
PRNSOL,U ! Write displacements
PRRSOL ! Write reactions
PRNSOL,S,PRIN ! Write principal streses
/OUTPUT ! Output redirection to the terminal
! _____Design Recapitulation
!***
/OUTPUT,%Job%.prn ! Output redirection into file
!
*VWRITE,'Hodnoten','ie spola','hlivosti','steny ','na stati','cke
zata','zenie '
(30X,8A8)
* VWRITE,' '
(1X,2A8)
* VWRITE,'Ru_lim =','Ru_max','Rc_lim =','Rs_min','Rt_lim
=','Rs_max
(22X,A8,E12.5,', ' ,15X,A8,E12.5,', ' ,15X,A8,E12.5)
* VWRITE,' '
(1X,2A8)
* VWRITE,'LC',Lc_,' Node =','Nu_max(1),' U_max
=','U_max(1),' Node =','Ns_min(1),' S_min =','S_min(1),' Node
=','Ns_max(1),' S_max =','S_max(1),
(1X,A2,F2.0,' ' ,A8,F5.0,' ' ,A8,E12.5,', ' ,A8,F5.0,'
',A8,E12.5,', ' ,A8,F5.0,' ' ,A8,E12.5)
* VWRITE,' '
(1X,2A8)
* VWRITE,'LC',Lc_,' Node =','Nu_max(1),'Ru_max
=','RFu_(1),' Node =','Ns_min(1),'Rc_min =','RFc_(1),' Node
=','Ns_max(1),'Rt_max =','Rft_(1),
(1X,A2,F2.0,' ' ,A8,F5.0,' ' ,A8,E12.5,', ' ,A8,F5.0,'
',A8,E12.5,', ' ,A8,F5.0,' ' ,A8,E12.5)
!
/OUTPUT ! Output redirection to the terminal
/SHOW,TERM ! Graf redirection to terminal
/SHOW,PNG ! Graf redirection into file Nbeamxx.png
/AUTO
/TRIAD,RBOT ! Define GSS display
/RGB,INDEX,100,100,100,0 ! Set black background
/RGB,INDEX,0,0,0,15
EPLOT ! Plot elements
!
/DSCALE,AUTO ! Set an automatic scaling
PLDISP,2 ! Plot original and deformed model
PLNSOL,S,X ! Plot stresses
PLNSOL,S,Y
PLNSOL,S,XY
PLNSOL,S,1 ! Plot principal stresses
PLNSOL,S,3

```



```
PLNSOL,S,INT
PLNSOL,S,EQV ! Plot Von Mises - stresses
PLVECT,S ! Plot stresses in vector forme

PLNSOL,EPEL,EQV ! Plot eqv. elastic strains
PLNSOL,EPPL,EQV ! Plot eqv. plastic strains
PLNSOL,EPTO,EQV ! Plot eqv. total strains
/SHOW,TERM ! Graf redirection to terminal
FINISH ! Finish Postprocesor
!.....
/RGB,INDEX,100,100,100,15 ! Set original color
/RGB,INDEX,0,0,0,0
EPLOT
*END ! Close editing mode from *CREATE
!*****
POST1N ! Run the macro Post1n.mac
/EXIT,NOSAV ! Exit task in Ansys software
```

Recapitulation of results :

$Ru_max = 1.25$, $Rc_min = 1.74$, $Rt_max = 0.51$

Fig. 10 Equivalent stress of I-Beam after nonlinear analysis

VII. CONTROL SYSTEM IN STRUCTURAL MODELLING

Significant project organizations have developed a quality program in accordance with the requirements of ISO 9001, which in the field of static calculations clearly defines the system of control of the used calculation models and tools.

We can control of input and output data:

A) By graphical control :

- color resolution of individual types of elements, geometric and material characteristics, local coordinate systems, output data,
- proportional display of element dimensions and their loads and results,
- symbolic representation of boundary conditions, interrelationships of elements and reactions.

B) By numerical control :

- control of listing of individual groups of input and output data (node coordinates, code numbers, geometric and physical characteristics, reactions, deformations and stress listing)

C) By logical control :

- control of input and output data in terms of their correct physical interpretation,
- checking the total weight of the model, the center of mass and the mass inertias of the model and the center of stiffness,
- control of output data in terms of their correct physical interpretation (proportionality, sign orientation, physical units,...),

- control of balance of forces from load and weight with reactions,
- checking the combination of individual load cases and their interpretation,
- control of deformations by comparison with a simplified solution.

VIII. CONCLUSION

This paper presents the experience of teaching FEM at the university. We currently have several software in FEM. Failure to know the nature of the problems leads to unprofessional and ineffective solutions to the static problem.

ACKNOWLEDGMENT

The project was realized with the financial support of the Grant Agency SR VEGA 1/0453/20.

REFERENCES

- [1] K. Weighard, Über Spalter und Zerrensen elastischer Körper, Zeithschr. für Math. Phys. 55 (1/2), 60-103, 1907.
- [2] M. Hazewinkel, (Ed.), *Encyclopedia of Mathematics on CD-ROM*, Kluwer, 1998, ISBN 0-7923-4805-2.
- [3] R. Courant, Variational methods for the solution of problems of equilibrium and vibrations, *Bulletin of the American Mathematical Society*, 49: 1-23. 1943, doi:10.1090/s0002-9904-1943-07818-4.
- [4] W. Ritz, Ueber eine neue Methode zur Lösung gewisser Variationsprobleme der mathematischen Physik, *J. Reine Angew. Math.*, 135 (1908) pp. 1-61, ISSN: 0075-4102; 1435-5345/e.
- [5] M. J. Turner, R. W. Clough, H.C. Martin, L.J. Topp, Stiffness and deflection Analysis of Complex structures, *J. Aeronautic Sci*, 23, 1956, doi:10.2514/83664.
- [6] R.W. Clough, Original formulation of the finite element method, *Finite Elements in Analysis and Design*, Volume 7, Issue 2, November 1990, Pages 89-101, doi.org/10.1016/0168-874X (90)90001-U.
- [7] O. C. Zienkiewicz, R. L. Taylor, *The Finite Element Method. Vol. 1 The Basis, Vol. 2 Solid Mechanics, Vol.3 Fluid Dynamics*, First ed. by McGraw-Hill 1967, Fift edit. pud. by Butterworth-Heinemann 2000, 2005., ISBN 0-7506-6320-0.
- [8] E. Hinton, B. Irons, Least squares smoothing of experimental data using finite elements, *Strain*, 4 (3): 24-27. 1968, doi:10.1111/j.1475-1305.1968.tb01368.x.
- [9] P.G.Ciarlet, P.A.Raviart, Maximum principle and uniform convergence for the finite element method, *Computer Methods in Applied Mechanics and Engineering*, Volume 2, Issue 1, February 1973, Pages 17-31, doi.org/10.1016/0045-7825(73)90019-4.
- [10] A. Jennings, *Matrix Computations for Engineerings and Scientists*, John Wiley, London 1977, 330p., ISBN 978-0471994213.
- [11] D.R.J. Owen, E. Hinton, *Finite Element Programming*, Academic Press, London, 604p., 1980, ISBN 0-906674-05-2.
- [12] A. H. Nilson, Z. Bažant, T. Y. Chang, et al. *Finite Element Analysis of Reinforced Concrete*, ASCE, New York, 1982.
- [13] J. Robinson, An evaluation of skew sensitivity of thirty-three plate bending elements in nineteen FEM systems, *Nuclear Engineering and Design*, Volume 90, Issue 1, 3 November 1985, Pages 67-85, doi.org/10.1016/0029-5493(85)90032-9.
- [14] T.J.R. Hughes, *Finite Element Method – Linear Static and Dynamic Finite Element Analysis*, Prentice Hall, Englewood Cliffs, 1987.
- [15] K.J. Bathe, *Finite Element Procedures*. Prentice Hall, New Jersey, 1065p., 1996, 2nd edit. 2014. ISBN 978-0-09790049-5-7.
- [16] T. Belytschko, J.Fish, B.E. Engelmann, A Finite Element with Embedded Localisation Zones, In.: *Computer Methods in Applied Mechanics and Engineering* 70, North-Holland, pp.59-89, 1988. D.Lasry, doi.org/10.1016/0045-7825(88)90180-6.
- [17] P. Kohnke, *ANSYS, Inc. Theory*, SAS IP Inc. Canonsburg, USA, ISO 9001: 1994 Company, 1266p. 2008.

- [18] E. Reissner, A note on variational principles in elasticity, *Int.J.Solids Structures*, 1, 93-95, 1995, doi.org/10.1016/0020-7683(65)90018-1.
- [19] L. Peng, J. Wang, Q. Zhu, Methods with high accuracy for finite element probability computing, *Journal of Computational and Applied Mathematics*, 59 (2), 181–189pp. 1995, doi:10.1016/S0045-7825(97)00021-2, doi:10.1016/0377-0427(94)00027-X.
- [20] J. P. Wolf, Ch. Song, *Finite-Element Modelling of Unbounded Media*, Edition John Wiley&Sons Ltd., Chichester, 1996, ISBN 0-471-961345.
- [21] Z. Bittnar, J. Šejnoha, *Numerical Methods in Structural Mechanics*, Telford, London, 470 pp. 1996. ISBN 0-7844-0170-5.
- [22] Ch. Song, J. P. Wolf, The scaled boundary finite-element method – alias consistent infinitesimal finite-element cell method – for elastodynamics, *Computer Methods in Applied Mechanics and Engineering*, 147 (3–4): 329–355. Bibcode: 1999 CMAME.147.329S. doi:10.1016/S0045-7825(97)00021-2.
- [23] A. Haldar, S. M. ahadevan, *Reliability Assessment Using Stochastic Finite Element Analysis*. John Wiley & Sons. pp. 344, 2000, ISBN: 978-0-471-36961-5.
- [24] E.L.Wilson, *Three-Dimensional Static and Dynamic Analysis of Structures, Computer and Structures*, Berkeley, 2002, ISBN 0-923907-00-9.
- [25] P. Šolin, K. Segeth, I. Doležel, *Higher-Order Finite Element Methods*, Chapman & Hall/CRC Press, 2003, ISBN 1-58488-438-X.
- [26] I. Babuška, U. Banerjee, J. E. Osborn, Generalized Finite Element Methods: Main Ideas, Results, and Perspective, *International Journal of Computational Methods*, 1 (1): 67–103. 2004, doi:10.1142/S0219876204000083.
- [27] J. N. Reddy, *An Introduction to the Finite Element Method* (Third ed.). McGraw-Hill. 2006, ISBN 9780071267618.
- [28] E. Madenci, I. Guven, *The Finite Element Method and Applications in Engineering Using ANSYS*, University of Arizona, Springer, 687p., 2006, ISBN: 8184897464.
- [29] T. Stolarski, Y. Nakasone, S. Yoshimoto, *Engineering Analysis with ANSYS Software*. Elsevier, Oxford, 2006, ISBN 0 7506 6875 X.
- [30] L.B. Veiga, F. Brezzi, A. Cangiani, G. Manzini, L.D. Marini, A. Russo, Basic principles of Virtual Element Methods, *Mathematical Models and Methods in Applied Sciences*, 23 (1): 199–214. 2013, doi:10.1142/S0218202512500492.
- [31] J. Králik, *Unilateral problems in the theory of elasto-plastic bodies*, PhD-dissertation work, FCE STU Bratislava, 1982.
- [32] J. Králik, M. Šimonovič, Earthquake Response Analysis of Nuclear Power Plant Buildings with Soil-Structural Interaction. *Mathematics and Computers in Simulation* 50. IMACS/Elsevier Science B.V, 1999, pp. 227-236, ISSN: 0378-4754.
- [33] J. Lovišek, J. Králik, Optimal Control for Elasto-Orthotropic Plate. *Control and Cybernetics*, 2006, No. 2, pp. 219-278, ISSN 0196-972235.
- [34] J. Králik, *Safety and Reliability of Nuclear Power Buildings in Slovakia. Earthquake-Impact-Explosion*. Monograph. Published by STU Bratislava, 2009, 307pp. ISBN 978-80-227-3112-6.
- [35] J. Králik, *Reliability Analysis of Structures Using Stochastic Finite Element Method*, Monograph. Published by STU Bratislava, 143p. 2009, ISBN 978-80-227-3130-0.
- [36] J. Králik, J. Králik, jr. Seismic Analysis of Reinforced Concrete Frame-Wall Systems Considering Ductility Effects in Accordance to Eurocode. *Engineering Structures*. Elsevier 2009, Vol.31, Issue 12, pp. 2865 - 2872, December 2009, ISSN 0141-0296.
- [37] J. Králik, Actual Problems of the Safety and Reliability of the NPP Structures in Slovakia, *Key Engineering Materials* (Volume 738) 2017, p. 261-272. doi.org/10.4028/www.scientific.net/KEM.738.261.
- [38] J. Králik, *Analysis of Structures – Finite Element Method*, STU Bratislava, 253 p., 2018. (in Slovak)

Contribution of Individual Authors to the Creation of a Scientific Article (Ghostwriting Policy)

Juraj Králik is profesor at the Department of Civil Mechanics, Faculty of Civil Engineering STU in Bratislava, he is an expert in safety and reliability of nuclear power plants and other structures.

Juraj Králik, jr. is an assistant at Institute of Structures in Architecture and Engineering Buildings, Faculty of Architecture and Design STU in Bratislava, he is an expert in computer mechanics and simulations.

Creative Commons Attribution License 4.0 (Attribution 4.0 International, CC BY 4.0)

This article is published under the terms of the Creative Commons Attribution License 4.0

https://creativecommons.org/licenses/by/4.0/deed.en_US