ISSN 1392 – 124X INFORMACINĖS TECHNOLOGIJOS IR VALDYMAS, 2004, Nr. 1(30)

COMPUTATIONAL TOOLS FOR SEMI-ANALYTICAL FINITE ELEMENT STABILITY ANALYSIS OF THIN-WALLED STRUCTURES

Michail Samofalov

Department of Strength of Materials, Vilnius Gediminas Technical University
Saulėtekio ave 11, LT-2040, Vilnius, Lithuania

Remigijus Kutas

Computer Center, Vilnius Gediminas Technical University
Saulėtekio ave 11, LT-2040, Vilnius, Lithuania

Rimantas Kačianauskas

Department of Strength of Materials, Vilnius Gediminas Technical University
Saulėtekio ave 11, LT-2040, Vilnius, Lithuania

Abstract. Computational tools implementing the semi-analytical finite elements for the linear and stability analysis of thin-walled beams are considered. The proposed method presents two-stage discretisation, where the thin-walled cross section is approximated by the semi-analytical finite elements at the first stage, while conventional longitudinal finite element discretisation is performed at the second stage. The developed computational tools for the processor allow assembling of the global linear and geometric stiffness matrices and present them in a form as it is used in conventional finite element software. The proposed computational tools for pre- and post-processors enable to perform communication between the user and convention finite elements software. Solution examples illustrate the use software tools and the quality of the results.

1. Indroduction

Investigation of the mechanical properties of various structures and their members continues to be a subject of a great interest in many fields of engineering. Different thin-walled structures, which have been increasingly used over the past few decades, belong to this category. Having a high ratio of stiffness and strength to the weight they belong to the most efficient load-carrying structural members, which are, however, highly susceptible to failure by instability [2, 6, 17, 18, 21, 33]. Design and analysis of the above structures require accurate analysis methods and practically simple computational tools.

The finite element method (FEM) [4, 8, 23, 35, 36] has been recognised for a long time as one of the most effective computational technology for analysing common structures under arbitrary loading and boundary conditions. The significant advances made in *finite element* (FE) technology are related not only with rapid development in computer hardware but also with the developments related to modelling strategies and programming concepts. The implementation of these concepts includes clear separation among computational and architectural components, standardisation of basic programming operations as a set of computational tools, standardisation of the interface between tools and a user etc. The tools share common date base and may be presented in the form of an independent executable modulus or utility subroutines. They are aimed at handling some specific functions such as date management, finite element library, standard mathematical operations, interactive procedures, computer graphics etc.

Description of the standard FE software may be found in almost every classical FE textbook [8, 23, 35] including edited in Lithuania [4, 9-11, 18]. A comprehensive review of the earliest FE software concepts is presented in [18, 26, 34]. The basic concepts are continuously held by the development of advanced widely used universal FE codes [1, 15, 24].

Advanced FEM codes have got a possibility to extend their capabilities due to the common newest developments in hardware and software technology. Besides computer graphics, the computer algebra systems such as VIBRAN [10, 11] developed in Lithuania or MATHEMATICA [22] and MATLAB [12] have obtained their rights, recently, as specific technological tools used for developing of the FE software. These techniques have got the advantage as they provide the symbolic mathematical models in the form of utility subroutines directly used in the software systems [9, 16, 25, 34].

Nowadays, not only universal but also specific application oriented FE codes are widely used in structural engineering [5, 13, 32]. The above software contains additional subject-oriented and user-friendly computational tools and/or utilities simplifying the use of software. There also exist FE codes developed for the thin-walled structures [27, 29].

Subsequently, this work deals with computational tools for the stability analysis of the thin-walled beams. The new advanced previously proposed *semi-analytical finite element method* (SAFEM) [3, 18, 19, 30, 31, 36] presents the thin-walled cross section as an assemblage of *semi-analytical finite elements* (SAFE).

A typical discretisation procedure may be considered as a standard computational technology comprising a mathematical model of the problem, the SAFE approach for thin-walled beams, algorithmic aspects of the linear and stability analysis and software developed.

The outline of the paper comprises specific software tools extending the possibilities of the standard FE analysis. Besides computational efficiency, the proposed method and computational tools developed give a structural designer the ability to simplify analysis and pre- and post-processing procedures reflecting mainly the geometry and topology of a thin-walled cross section. Some examples illustrate the usage of the SAFEM and software.

2. Mathematical models and concept of semi-analytical finite elements

2.1. Mathematical models

The formulation of a stability problem as well as corresponding expressions for stresses, strains and other aspects are widely discussed in [6, 10, 16, 17, 19, 21, 24, 29, 31, 32, 34]. A critical condition, at which instability of a structure occurs, is obtained considering the second variation of the total potential energy.

Formulation of the numerical mathematical models to be applied for the structural analysis requires the continuous variables to be expressed in terms of the algebraic nodal displacements. Finally, the linearised mathematical model is expressed as an algebraic eigenvalue problem [20]:

$$\left(\left[\boldsymbol{K}\right]-\lambda_{cr}\left[\boldsymbol{K}_g\right]\right)\boldsymbol{U}=\boldsymbol{0}\,, \tag{1}$$

where $[\,\boldsymbol{K}\,]$ is a linear stiffness matrix, $[\,\boldsymbol{K}_g\,]$ is a geometric stiffness matrix, eigenvector $\boldsymbol{U}$ is a vector of nodal displacements representing buckling mode shape, while eigenvalue λ_{cr} is a critical load factor defined as the ratio of the critical load $\boldsymbol{F}_{cr}$ with respect to a given load parameter of the external loads $\boldsymbol{F}$:

$$\boldsymbol{F}_{cr}=\lambda_{cr}\,\boldsymbol{F}\,. \tag{2}$$

The simplest *linear static analysis* problem, solution of which is required for the evaluation of geometric stiffness matrix may be expressed in terms of displacements and presented as:

$$\left[\boldsymbol{K}\right]\boldsymbol{U}=\boldsymbol{F}\,, \tag{3}$$

where $\boldsymbol{U}$ is a vector of unknown nodal displacements and $\boldsymbol{F}$ is a given vector of nodal forces.

Evaluation of global models (1) and (3) containing boundary conditions is performed by assembling of local subdomains depending on the approximation techniques applied.

2.2. A concept of semi-analytical finite elements

A concept of the SAFEM [3, 19, 30, 31] will be presented in the following manner. The considered beam can be represented as a composition of thin walls (slender rectangular plates) interconnected along their longitudinal edges (Figure 1). Each of the wall segments has got an individual height h, thickness t, Young's E and shear G modulus. The thickness is assumed to be very small in comparison with the height of the SAFE, thus $t \ll h$.

The beam is described by longitudinal co-ordinate Ox and cross-sectional co-ordinates Oy and Oz. For the sake of convenience, the global co-ordinates y and z are replaced by single perimetric co-ordinate $p = p(y, z)$.

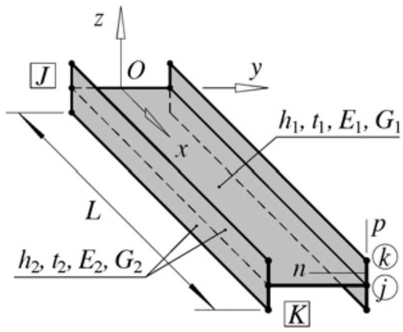

Figure 1. Illustration of the thin-walled beam and discretisation concept

The cross section built up of straight segments is assumed to be a plane frame assembled by one-dimensional elements. Such FE, used for description of the cross-sectional variation of field variables is termed here as a SAFE. The position of single SAFE is locally defined by two nodes j and k.

The fragment of the beam having length L and replaced between the two global nodes J and K may be considered as a complex conventional FE, while, the wall segment, the cross section of which is defined by a single SAFE, is termed as a *subelement* (SE).

The proposed semi-analytical discretisation technique of thin-walled beams is a two-stage procedure comprising discretisation of geometry and three-dimensional displacement field. The main objective of *the first stage* is discretisation of a thin-walled cross-section. Within individual SAFE denoted by a subscript *safe* one-dimensional approximation of displacements $\boldsymbol{u}$ is to be assumed:

$$\boldsymbol{u}_{safe}(x,p)=[\,\boldsymbol{f}(p)\,]\boldsymbol{U}_{safe}(x), \tag{4}$$

where $[\,\boldsymbol{f}(p)\,]$ is a cross-sectional approximation matrix and $\boldsymbol{U}_{safe}(x)$ is a vector of generalised displacements usually attached to the cross-section nodes (generatric axis).

The second stage is applied for longitudinal discretisation by using standard FEM procedures. Longitudinal variation of generalised displacements within the subelement is presented as

$$\boldsymbol{U}_{safe}(x)=[\,\boldsymbol{N}(x)\,]\boldsymbol{U}_{se}\,, \tag{5}$$

where $[\,\boldsymbol{N}(x)\,]$ is a matrix of displacement shape functions well known in the FEM.

Hence, the conventional FE of thin-walled beams is presented as a complex one-dimensional FE assembling of SE, while the entire beam structure is assembled of SE. The hierarchy of discretisation is presented in Figure 2.

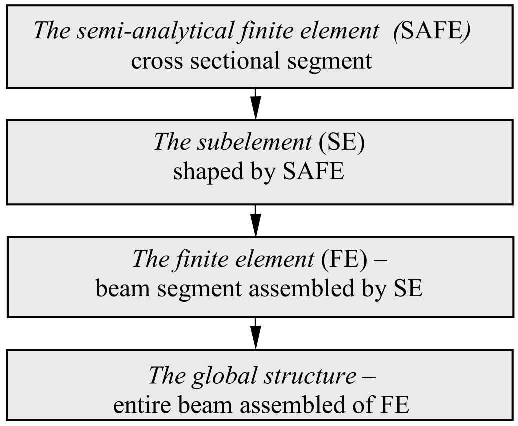

Figure 2. Hierarchy of SAFE discretisation of the thin-walled beams

The three-dimensional strain and stress fields may be approximated in a similar way as displacements (4)-(5). The strains:

$$\boldsymbol{\varepsilon}_{safe}(x,p)=[\,\boldsymbol{F}(p)\,][\,\boldsymbol{B}(x)\,]\boldsymbol{U}_{se}\,, \tag{6}$$

where $[\,\boldsymbol{F}(p)\,]$ is a cross-sectional while $[\,\boldsymbol{B}(x)\,]$ is a longitudinal strain approximation matrices obtained by differentiation of the matrices $[\,\boldsymbol{f}(p)\,]$ and $[\,\boldsymbol{N}(x)\,]$, respectively.

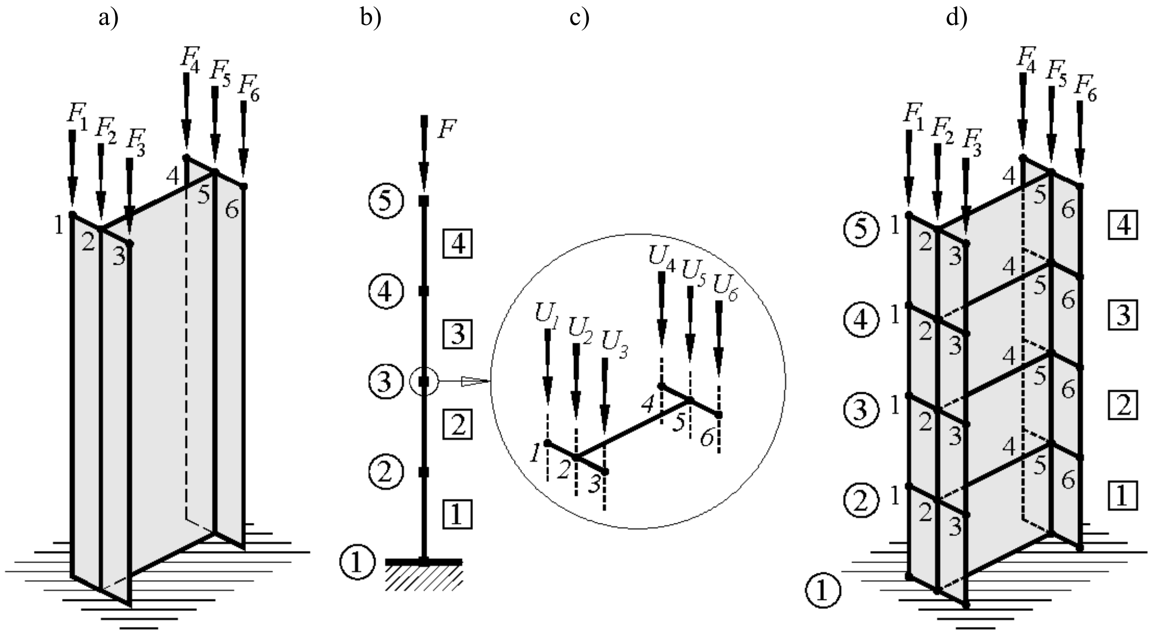

Figure 3. A thin-walled cantilever beam under longitudinal loading: general view (a), global model assembled of one-dimensional FE (b), model of global node (cross-section) assembled of SAFE (c) and global model assembled of two-dimensional subelements

A thin-walled cantilever beam under longitudinal loading (Figure 3) is presented as an example for illustration of discretisation. The global schematic model presents a structure assembled of one-dimensional FE (Figure 3b), which is presented in a form used in conventional structural analysis. The main differences occur in construction of the global nodes (Figure 3c), model of which presents the cross section assembled of SAFE. Here, local *degrees of freedom* (DOF) are referred to local nodes. Actually, the global model means a composition of SE (Figure 3d).

2.3. Characteristic matrices of subelement

Formulation of the mathematical model (1) requires calculation of the two global characteristic matrices [$\boldsymbol{K}$] and [$\boldsymbol{K}_g$] reflecting properties of individual finite elements and subelements. As it is obvious in the FEM, this standard procedure starts from the derivation of characteristic matrices on the lowest level. By applying the above approximations (4) and (6), the linear stiffness matrix of an individual SAFE is presented as follows:

$$\left[\boldsymbol{K}_{safe}(x)\right]=\int_A \left[\boldsymbol{F}(p)\right]^{\mathrm{T}}\left[\boldsymbol{D}\right]^{-1}\left[\boldsymbol{F}(p)\right]\mathrm{d}A\,, \qquad (7)$$

where A is a cross-sectional area of a rectangular segment, [$\boldsymbol{D}$] is an elasticity matrix of the continuum.

Now, the linear stiffness matrix of higher-level subelement may be obtained in the same way as:

$$\left[\boldsymbol{K}_{se}\right]=\int_L \left[\boldsymbol{B}(x)\right]^{\mathrm{T}}\left[\boldsymbol{K}_{safe}(x)\right]\left[\boldsymbol{B}(x)\right]\mathrm{d}x\,. \qquad (8)$$

The same procedures are used for derivation of the geometric stiffness matrix. For SAFE

$$\left[\boldsymbol{K}_{g\,safe}\right]=\int_A \left[\boldsymbol{F}_g(p)\right]^{\mathrm{T}}\left[\boldsymbol{s}(x,p)\right]\left[\boldsymbol{F}_g(p)\right]\mathrm{d}A \qquad (9)$$

and for subelement:

$$\left[\boldsymbol{K}_{g\,se}\right]=\int_L \left[\boldsymbol{B}_g(x)\right]^{\mathrm{T}}\left[\boldsymbol{K}_{g\,safe}(x)\right]\left[\boldsymbol{B}_g(x)\right]\mathrm{d}x\,. \qquad (10)$$

Here, [$\boldsymbol{F}_g(p)$] and [$\boldsymbol{B}_g(x)$] are matrices obtained by differentiation of shape functions [$\boldsymbol{f}(p)$] and [$\boldsymbol{N}(x)$], while [$\boldsymbol{s}(x, p)$] is the matrix composed of longitudinal stress components.

2.4. Assembling technique

Final computation of the global matrices in expression (1) has to be performed using the assembling procedure. This is a standard FEM procedure mathematically expressed as a congruent transformation. The characteristic matrix of the FE may be presented in the following form:

$$\left[\boldsymbol{K}_{fe}\right]=\sum\left[\boldsymbol{G}_{se}\right]^{\mathrm{T}}\left[\boldsymbol{K}_{se}\right]\left[\boldsymbol{G}_{se}\right], \qquad (11)$$

where [$\boldsymbol{G}_{se}$] is the incidence matrix for each SE, reflecting topological properties of an individual SE.

The global characteristic matrix of the entire beam structure is calculated in the same manner by assembling FE:

$$\left[\boldsymbol{K}\right]=\sum\left[\boldsymbol{G}_{fe}\right]^{\mathrm{T}}\left[\boldsymbol{K}_{fe}\right]\left[\boldsymbol{G}_{fe}\right]. \qquad (12)$$

It is necessary to note that the standard kinematical boundary conditions with assumed zero displacement are incorporated through the incidence matrices.

Assembling relations (11)-(12) should finally provide the global models (1)-(3) defined in terms of the global characteristics. Actually, it is merely on algorithmic double summation procedure running through the characteristic matrices of individual SE and FE, implementation of which will be presented below.

3. Computation algorithm

3.1. Principal sequence

Development of a computational algorithm presents an important step of modelling resulting in the quality of computer implementation of the SAFEM. The FEM has advantage, that it is implemented using sequential modular approach, according to which the computational process at a certain level is subdivided into relatively independent standard sub-processes or modules. As a result, the FE algorithm shaping the software architecture can be designed as a sequence of the principal operations reflecting the type of problem-dependent FE analysis. A typical sequence of the algorithm comprises the operations of linear analysis adapted to more complex problems. The sequence of standard operations describing the stability analysis problem (1) is illustrated in Figure 4.

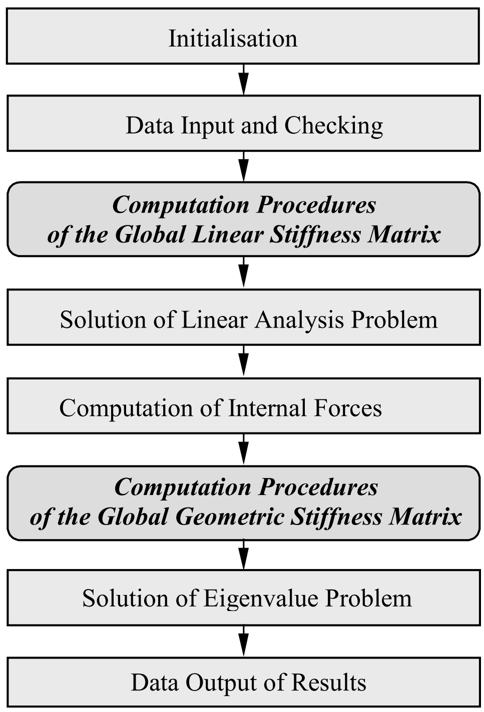

Figure 4. General algorithm of FE stability analysis

The initialisation phase generates the program configuration and initiates the database. Data managing is aimed to generate and check of initial data describing geometry and physical properties of the structure, topology of FE system, external loading etc. Computation procedures of the global linear stiffness matrix of the whole structure comprise generation, transformation and assembling operations of the individual FE stiffness matrices. This step presents a problem-dependent part related to the particular problem through the characteristic matrices of the particular FE. Solution of linear algebraic equations (3) comprises standard mathematical operations providing the values of the nodal displacements as primary variables. Computation of the secondary variables in the process of the structural analysis allows providing calculation of the internal forces and stresses.

The above operations describe the standard sequence of linear analysis and are an obligatory part of any FE algorithm, while the next two blocks present additional operations of the stability analysis. Computational procedures of the global geometric stiffness matrix are the same as for the linear stiffness matrix, but in this case, secondary variables are used for matrix generation. The next step means the formal solution of the standard eigenvalue problem (1). The last operations compute the final values of numerical results and present them in a graphical format.

The main differences of the proposed SAFEM lie in the computation procedures of the global linear and geometric stiffness matrices.

3.2. Computation of the global stiffness matrices

In the conventional FE method computation procedure of the global stiffness matrices [$\boldsymbol{K}$] as well as [$\boldsymbol{K}_g$] is described by expression (12) and presents a single loop consisting of four standard steps for each of the FE. They are generation of the stiffness matrix [$\boldsymbol{K}_{fe}$] with original and physical properties, formation of the incidence matrix [$\boldsymbol{G}_{fe}$] using logical rules, congruent transformation according to expression (12) and incorporation into the global stiffness matrix.

In the proposed SAFE approach the computation procedures of global matrices [$\boldsymbol{K}$] and [$\boldsymbol{K}_g$] are described by two models (11) and (12) and have got two assembling loops, the first of which (here termed as external one) manages a standard assembling of FE, whereas the second (termed here as internal one) assembling loop describes generation of individual stiffness matrix for each subelement (Fig. 5).

The internal loop implements assembling of FE stiffness matrix [$\boldsymbol{K}_{fe}$] by (11) in a standard FEM manner and serves as the base for the conventional assembling (12). The proposed algorithm has advantage, that it deals actually with standard operations.

These procedures are similar for linear and geometric stiffness matrices as well as for the external load vector.

3.3. Formation of the incidence matrices

The topological properties of the thin-walled beam structure as a whole are described by the incidence matrices for the FE [$\boldsymbol{G}_{fe}$] and SE [$\boldsymbol{G}_{se}$], respectively. Since the formation of the conventional FE matrices is well known in the literature, let us focus on the algorithm of formation of topological description of a cross section.

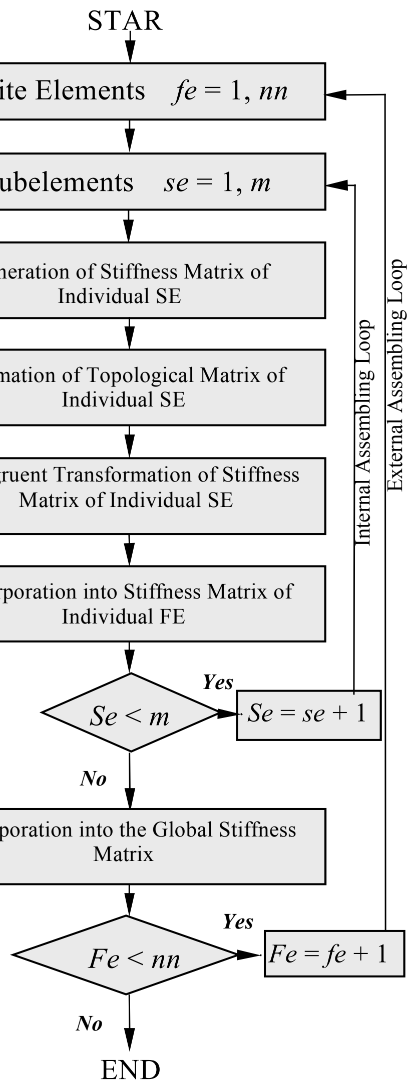

Figure 5. General algorithm for assembling of the global matrix

The incidence matrix is a Boolean's matrix projecting one vector onto another by means of logical transformation. The incidence matrix [$\boldsymbol{G}_{se}$] of the SE shapes the vector of nodal variables of the SE $\boldsymbol{U}_{se}$ into the vector of the entire FE:

$$\boldsymbol{U}_{fe} = \left[\boldsymbol{G}_{se}\right]\boldsymbol{U}_{se} \,. \tag{13}$$

Actually, the vectors $\boldsymbol{U}_{fe}$ and $\boldsymbol{U}_{se}$ present composition of the nodal variables in two global nodes J and K, (Figure 1). These vectors may be presented as

$$\boldsymbol{U}_{fe} = \left\{ \boldsymbol{U}_{feJ} \quad \boldsymbol{U}_{feK} \right\}^{\mathrm{T}} \text{ and}$$

$$\boldsymbol{U}_{se} = \left\{ \boldsymbol{U}_{seJ} \quad \boldsymbol{U}_{seK} \right\}^{\mathrm{T}} .$$

The incidence matrix [$\boldsymbol{G}_{se}$] of the SE may be decomposed in the same manner and is presented as a block diagonal matrix

$$\left[\boldsymbol{G}_{se}\right]=\begin{bmatrix}\boldsymbol{G}_{seJ} & \boldsymbol{O}\\ \boldsymbol{O} & \boldsymbol{G}_{seK}\end{bmatrix}.$$

Now it is clear, that instead of dealing with full matrices and vectors, it is reasonable to consider a single global node. Denoting $\overline{\boldsymbol{U}}_{fe} \equiv \boldsymbol{U}_{feK}$, $\overline{\boldsymbol{U}}_{se} \equiv \boldsymbol{U}_{seK}$ and $\left[\overline{\boldsymbol{G}}_{se}\right] \equiv \left[\boldsymbol{G}_{seK}\right]$, consideration new topology transformation (13) may be simply replaced by consideration of new transformation reflecting topology of the entire cross section

$$\overline{\boldsymbol{U}}_{fe}=\left[\overline{\boldsymbol{G}}_{se}\right]\overline{\boldsymbol{U}}_{se}, \tag{14}$$

where $\overline{\boldsymbol{U}}_{se}$ and $\left[\overline{\boldsymbol{G}}_{se}\right]$ mean nodal displacements and incidence matrix of individual SAFE, while $\overline{\boldsymbol{U}}_{fe}$ stands for cross-sectional displacements.

The description of the topology of a cross section is illustrated by using a representative example. The typical thin-walled SE defined by two global nodes J and K and two local nodes j and k and its DOF are presented in Figure 6. The DOF of global node K presents DOF of single SAFE incorporated into vector $\overline{\boldsymbol{U}}_{se}$. This SAFE is applied for discretisation of I-section (Figure 7).

The thin-walled SE (Figure 6) is constructed as a plate having longitudinal and transverse membrane DOF and distortional (bending) DOF [17, 30, 31]. The main difficulty in formation of a relatively simple assembling procedure is the conformity of the different displacement components approximated by different laws.

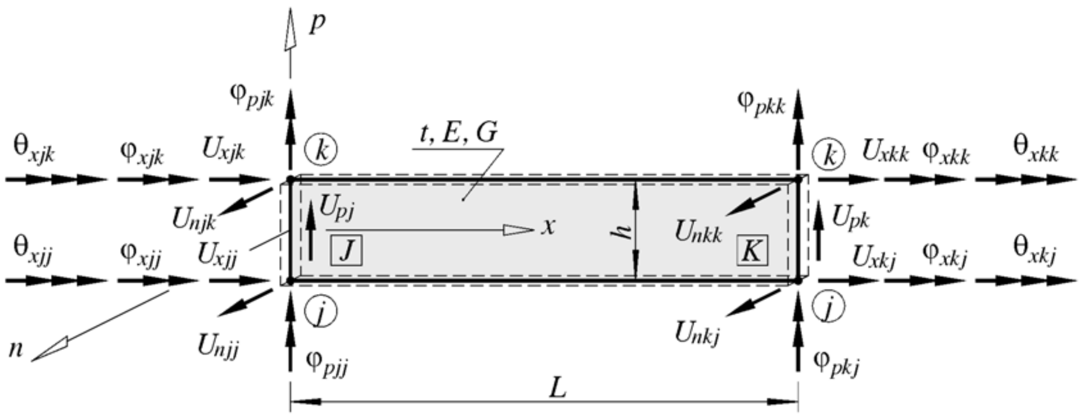

Figure 6. The subelement and its degrees of freedom

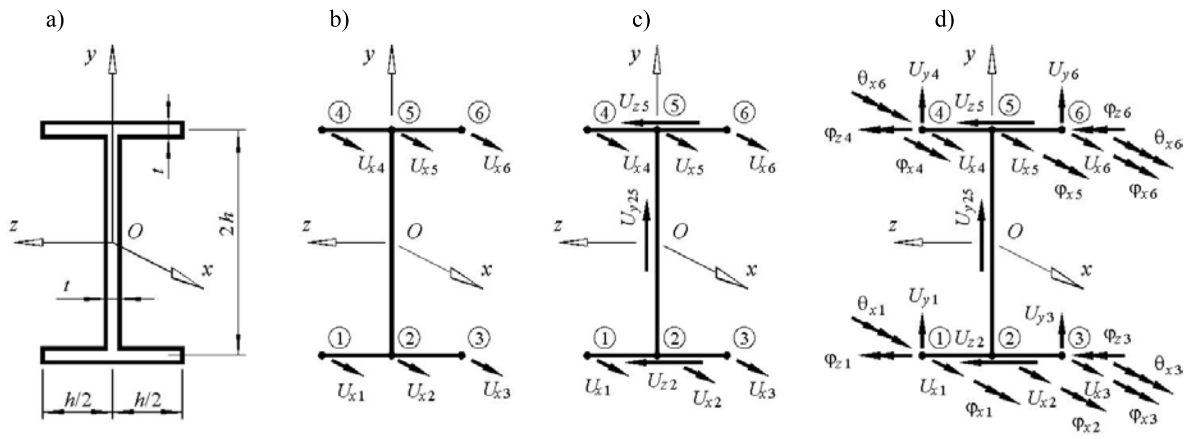

Figure 7. Discretisation of I-section by semi-analytical finite elements: section geometry (a); longitudinal membrane DOF (b); longitudinal and transversal membrane DOF (c); membrane and distortional DOF (d)

Since longitudinal membrane displacements vary linearly along a perimetric co-ordinate, the required C_0 continuity may be simply guaranteed by stacking of longitudinal DOF, while the membrane transverse displacements being constant in SAFE, remain incompatible.

The distortional transversal displacements are approximated by the third-order polynomials and require C_1 continuity along the edges. The continuity of longitudinal distortional rotations may be simply fulfilled by stacking nodal rotations φ_x. Due to a different order of approximation of transverse membrane u_p and distortion u_n displacements, it is difficult to impose a full C_1 continuity along the entire connection edge of a thin-walled beam. Therefore, a simplified linear longitudinal approximation of displacements $u_n(x)$ and rotations $\varphi_p(x)$ neglecting their derivatives is assumed throughout the current investigation.

For the sake of simplicity, the topology of a cross section is described in a hierarchical order. Let us start by membrane DOF only. As an example, the simplest model of *I*-section (Fig. 7b) is composed of five SAFE relating local nodes 1-2, 2-3, 2-5, 4-5 and 5-6.

The vector of sectional displacements is now defined by six nodal components as $\overline{\boldsymbol{U}}_{fe} = \{U_{x1},\ U_{x2},\ U_{x3},\ U_{x4},\ U_{x5},\ U_{x6}\}^{\mathrm{T}}$, while displacements vectors of individual SAFE as $\overline{\boldsymbol{U}}_1 = \{U_{x1},\ U_{x2}\}^{\mathrm{T}}$, $\overline{\boldsymbol{U}}_2 = \{U_{x2},\ U_{x3}\}^{\mathrm{T}}$, $\overline{\boldsymbol{U}}_3 = \{U_{x2},\ U_{x5}\}^{\mathrm{T}}$, $\overline{\boldsymbol{U}}_4 = \{U_{x4},\ U_{x5}\}^{\mathrm{T}}$ and $\overline{\boldsymbol{U}}_5 = \{U_{x5},\ U_{x6}\}^{\mathrm{T}}$. The incidence matrices of the above SAFE now are as follows:

$$\left[\overline{\boldsymbol{G}}_1\right]=\begin{bmatrix}1&0\\0&1\\0&0\\0&0\\0&0\\0&0\end{bmatrix};\quad \left[\overline{\boldsymbol{G}}_2\right]=\begin{bmatrix}0&0\\1&0\\0&1\\0&0\\0&0\\0&0\end{bmatrix};\quad \left[\overline{\boldsymbol{G}}_3\right]=\begin{bmatrix}0&0\\1&0\\0&0\\0&0\\0&1\\0&0\end{bmatrix};$$

$$\left[\overline{\boldsymbol{G}}_4\right]=\begin{bmatrix}0&0\\0&0\\0&0\\1&0\\0&1\\0&0\end{bmatrix};\quad \left[\overline{\boldsymbol{G}}_5\right]=\begin{bmatrix}0&0\\0&0\\0&0\\0&0\\1&0\\0&1\end{bmatrix}.$$

Topology of the cross section defined by the above matrices may be practically presented as a single topological vector, every member of which points to the number of the corresponding DOF:

$$\boldsymbol{V}_1=\{\ 1\quad 2\quad 2\quad 3\quad 2\quad 5\quad 4\quad 5\quad 5\quad 6\ \}^{\mathrm{T}}.$$

The above methodology may be simply extended to incorporate on additional DOF. In the case of transversal DOF, the local orientation of DOF is of major importance, the orientation may be defined by an algebraic sign. When the direction of the local axis *p* is negative, the number of topological DOF will be negative too.

The next example illustrates *I*-section with longitudinal and transversal DOF (Fig. 7c): The vector of cross-sectional displacements is now defined by nine nodal components – six longitudinal and three independent transversal components as $\overline{\boldsymbol{U}}_{fe} = \{U_{x1},\ U_{x2},\ U_{x3},\ U_{x4},\ U_{x5},\ U_{x6},\ U_{y25},\ U_{z2},\ U_{z5}\}^{\mathrm{T}}$, while displacements vectors of individual SAFE as $\overline{\boldsymbol{U}}_1 = \{U_{x1},\ U_{x2},\ U_{z2}\}^{\mathrm{T}}$, $\overline{\boldsymbol{U}}_2 = \{U_{x2},\ U_{x3},\ U_{z2}\}^{\mathrm{T}}$, $\overline{\boldsymbol{U}}_3 = \{U_{x2},\ U_{x5},\ U_{y25}\}^{\mathrm{T}}$, $\overline{\boldsymbol{U}}_4 = \{U_{x4},\ U_{x5},\ U_{z5}\}^{\mathrm{T}}$ and $\overline{\boldsymbol{U}}_5 = \{U_{x5},\ U_{x6},\ U_{z5}\}^{\mathrm{T}}$.

The topological vector of the entire cross section may be presented as

$$\boldsymbol{V}_2=\{\ 1\quad 2\quad -8\quad 2\quad 3\quad -8\quad 2\quad 5\quad 7\quad 4\quad 5\quad -9\quad 5\quad 6\quad -9\ \}^{\mathrm{T}}.$$

For the most general case including distortion (Figure 7d), the vector of cross-sectional displacements is now defined by adding 27 nodal distortional components $\overline{\boldsymbol{U}}_{fe} = \{\ U_{x1},\ U_{x2},\ U_{x3},\ U_{x4},\ U_{x5},\ U_{x6},\ U_{y1},\ U_{y25},\ U_{y3},\ U_{y4},\ U_{y6},\ U_{z2},\ U_{z5},\ \varphi_{x1},\ \varphi_{x2},\ \varphi_{x3},\ \varphi_{x4},\ \varphi_{x5},\ \varphi_{x6},\ \varphi_{z1},\ \varphi_{z3},\ \varphi_{z4},\ \varphi_{z6},\ \theta_{x1},\ \theta_{x3},\ \theta_{x4},\ \theta_{x6}\ \}^{\mathrm{T}}$.

Finally, the topological vector describing all 55 DOF of the full matrix is as follows:

$$\boldsymbol{V}_3=\{\begin{array}{ccccccc}1&2&-12&7&14&-20&24\\9&15&0&0&2&3&-12\\9&15&0&0&8&16&-21\\25&2&5&9&12&15&0\\0&13&18&0&0&4&5\\-13&10&17&-22&26&9&18\\0&0&5&6&-13&9&18\\0&0&11&19&-23&27\end{array}\}^{\mathrm{T}}.$$

For the automatic generation of the topology vector the following algorithm is suggested. All nodes of a thin-walled cross section may be classified as external or internal ones. The external nodes are, actually, free-end nodes and contain all six DOF. The internal nodes join two or more neighbour SAFE. In this case, all longitudinal DOF are general for each SAFE, but transversal DOF may be parallel or orthogonal. However, the orthogonal DOF are not connected. The formation algorithm of the cross-sectional topological vector is illustrated in Figure 8.

4. Software implementation

4.1. Software concept

Software implementation is the most important phase in the development of computational tools for each mathematical analysis problem. This is an expensive and time-consuming task requiring both the ingenuity of those who carry it out and knowledge of strong scientific disciplines.

Flexibility of computing systems is the major feature mandatory for design and implementation of the software.

A flexible software system should be designed with an idea that it will be modified and its architecture should take into account future changes. This is provided by the modularisation concept and software tools. The software tools present independent executable modulus or utility subroutines to handle specific computational procedures. Most of them are internal modulus programmed by a software designer, while some of them may be written by a user employing original internal or independent programming languages [14, 23, 24, 34].

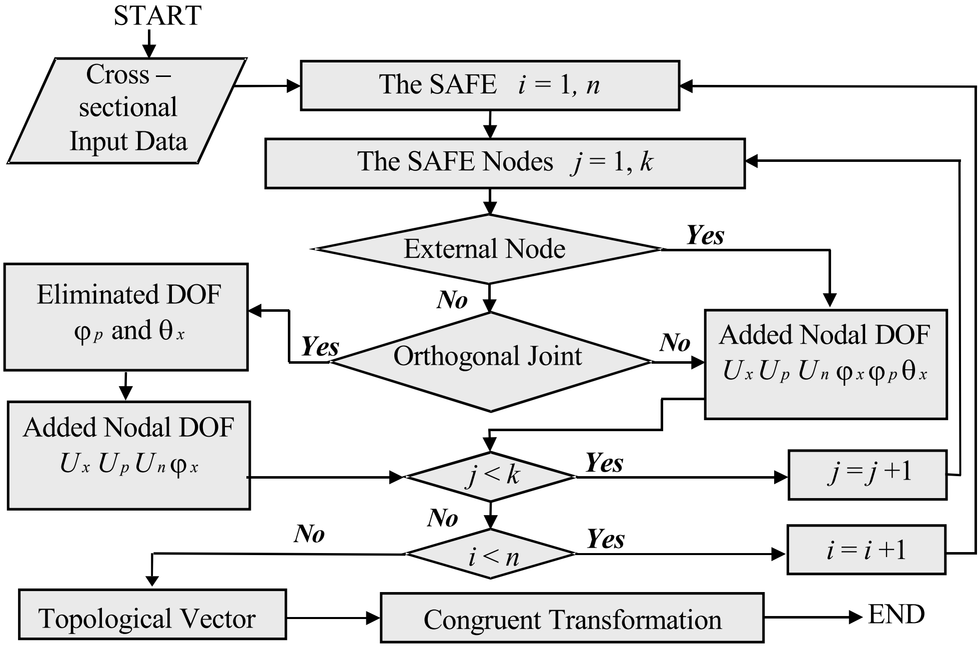

Figure 8. The formation algorithm of the cross-sectional topological vector

A wide variety of the existing finite element codes, for example, ANSYS [1], NASTRAN [24] etc., are modular software systems. A typical structure of the FEM software (Fig. 9) comprises three basic parts: pre-processor, processor and post-processor. The function of the pre-processor is to prepare the input data and arrange them according to a format required by the processor, the function of processor is to form and solve a particular mathematical model, while the function of the post-processor is to compute the output results, store or display them in the format required by a user.

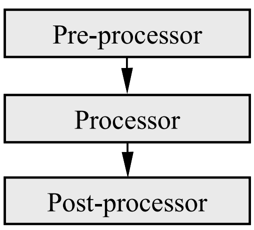

Figure 9. A typical structure of the FEM software

The SAFEM discretisation technique being applied to stability analysis of thin-walled structures is implemented as a software system SATW. The program SATW consists of three basic blocks and an additional part for generation of SE characteristic matrices. The first block creates a new data file and generates SAFE mesh for one-dimensional FE system. The next block, using a model of SE from the additional part, starts the assembling procedure of global characteristic matrices and other standard solutions. The last block of the program presents the numerical results using graphical interface and additional text files, which comprise information about deformed shape, stress state and about two of the most important stability parameters: critical value of the external load and critical shape modes.

Actually, it presents modification of standard modular FE software accomplished by introducing of new computational tools. The graph of the software designed for stability analysis is presented in Figure 10, where new tools reflecting the most important features of the SAFEM are accentuated. Below we discuss the details of the implementation of the software tools developed on the basic algorithm, which has been presented in the previous section.

4.2. Software tools for pre-processor

The topological properties of a thin-walled cross section are automatically determined by the presented algorithm (Figure 8) of the program SATW. The cross section is defined independently from the co-ordinate system centre and orientation of axes. The SAFE and the nodes are arbitrarily numbered. Initially, the program starts with a SAFE and analyses each subnode of it. In case of an orthogonal joint, rotation and warping of the SAFE are eliminated through all global nodes and finite elements. As a result, the topological vector is obteined and may be successfully used for congruent transformation (11) of the cross-sectional DOF of the linear stiffness and geometric stiffness matrices.

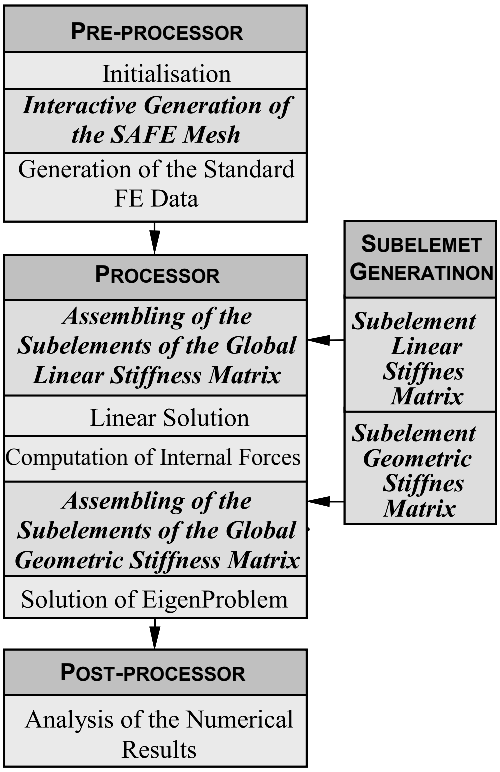

Figure 10. Graph of the modular software for stability analysis by the SAFEM

4.3. Software tools for processor

The program SATW is developed on the basic idea of the modular software system [8]. Three basic modules can be independently used and modified for special tasks. There are two differences of the program SATW: new characteristic matrices as well as new models with additional DOF can be incorporated in all blocks; the new algorithm for assembling procedures of the linear stiffness matrix and geometric stiffness matrix is used. The blocks are written in Fortran [14], while an additional part of the program is programmed using the standard package of computer algebra MATHEMATICA [22].

The symbolic manipulations in expressions (7)-(10) of mathematical integration and other analytical operations are automatically performed and the results are presented in a general algebraic form. Symbolic expressions of subelement linear stiffness and geometric stiffness matrices are generated by the additional part of the program and presented as two Fortran-type independent subroutines. The use of computer algebra makes an analytical solver part of SAFEM analysis as fast as possible.

Another specific software tool is *assembling* tool developed for computation of the global matrices by assembling of subelements. This tool implements the algorithm illustrated in Figure 5 and replaces a conventional assembling procedure. Here, the results of previously described software tools are presented. Instead of dealing with FE the assembling tool employs SE subroutines generated by the SE generator.

Instead of dealing with congruent transformations (11)-(12), assembling is implemented as a direct summation of FE matrices, where topological information held by incidence matrix $[\,\boldsymbol{G}_{fe}\,]$ is stored in the topological vector.

Finally, a direct summation of SE matrices is obtained by combining the topology of conventional FE and topology of cross section. This information is stored in the two topological vectors, namely, in conventional FE topology vector obtained in a standard manner and cross-sectional topology vector produced by previously described software tool for pre-processor.

The *assembling* tool is used initially for computation of the global $[\,\boldsymbol{K}\,]$ linear stiffness matrix. After computation of stress values as secondary variables, it is also used for computation of the geometric stiffness matrix $[\,\boldsymbol{K}_g\,]$.

4.4. Software tools for post-processor

The post-processor block transforms given results to text and graphical results of a real mechanical problem. The graphical package draws the FE system as an assemblage of nodal points connected by lines. The linear and non-linear computational results are shown by deformed shape of the middle surface of a thin-walled beam. This graphical module is used as “black box”, managed by the input data and not modified.

5. Numerical examples

Various examples of *I*-section beams under longitudinal loading have been tested to verify the proposed SAFE approach and, finally, suitability of the proposed software tools. The instability behaviour of thin-walled beams is very sensitive to various boundary conditions and possible imperfections, which may provide different buckling modes. In general, the classical theory of thin-walled beams [33] assumes an undistorted cross section. Analytical solutions developed on this base and the corresponding FE [6, 21] are sufficient for predicting the global buckling.

In case of predominating distortion and local buckling, the shell FE model is a single tool suitable for non-linear stability analysis. In our case, it is used for verification purpose and is treated here as an exact one. The large number of the DOF increases the demand of computer memory and the time costs and is the most significant obstacle restricting application of this approach.

The general purpose ANSYS code [1, 23] have been used for the sake of comparison. Here, the shell

FE is defined by four nodes with six DOF (three translation and three rotation) on each edge.

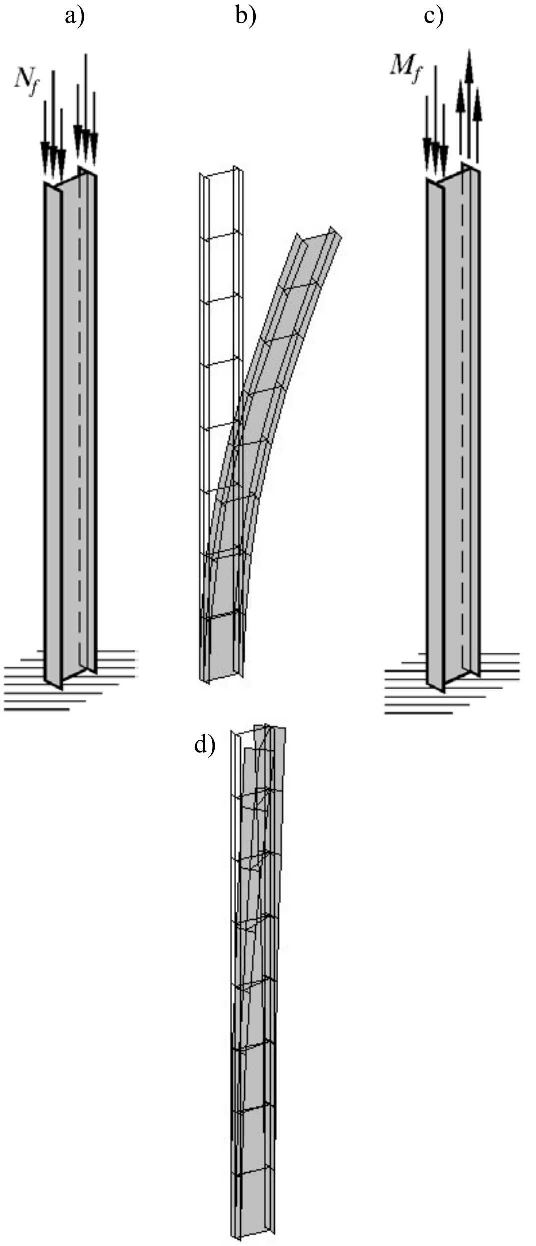

Figure 11. The beam with constant *I*-section: structure under the axial force (a) and its flexible buckling mode (b); structure under bending moment (c) and its torsion-flexible buckling mode (d)

In *the first example*, the cantilever beam with constant *I*-section acting by axial force and bending moment is presented for verification of the stability. The geometry of the beam is relatively expressed by the characteristic cross-sectional parameter h, while the entire length of the beam and the thickness of its cross-sectional wall are defined as $L = 40h$ and $t = 0{,}02h$. The properties of a homogeneous elastic material are defined by Young's and shear modulus $E/G = 2.5$. The beam is acted by an axial load and bending moments, which are expressed as a combination of longitudinal nodal loads at the flanges F_1, F_2, F_3 and F_4, F_5, F_6. The beam is considered as an assemblage of the five one-dimensional SAFE defined by six nodes (Figure 11).

On the basis of given results the conclusion have been drawn about the agreement of the proposed model with the classic analytical theory (Table 1).

The second example investigates the local stability of the column with variable *I*-section under an combined action of an axial force and plane bending moment (Figure 12a). The relative height of the cross-sectional web is expressed by relation $a/b = 1$ and $a/b = 2$ (constant and variable cross sections), where a and b are the height at the ends of the column.

Table. 1. The results of the constant section solutions

FE model	Characteristics		
	FE	DOF	λ_{cr}
Classic Theory	—	—	1,00
Axial Force			
Shell FE	384	2574	0,99
SAFE	32	864	0,99
	16	432	0,99
	8	216	0,98
Bending			
Shell FE	384	2574	0,99
SAFE	32	864	0,99
	16	432	0,98
	8	216	0,97

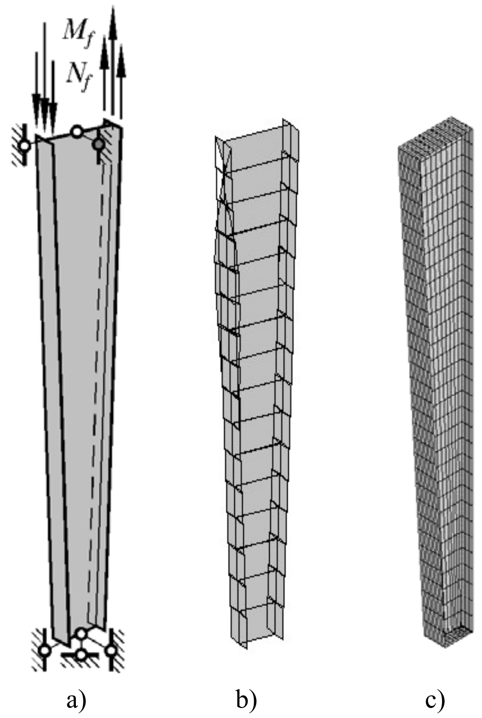

Figure 12. The column with variable *I*-section (a) and its local buckling mode (b); column's solution results by shell FE (c)

The critical state is obtained by numerical experiments using the SAFE (Figure 12b) and shell FE (Figure 12c). The column with variable *I*-section is presented as assemblage of one-dimensional FE with a constant cross section. Each node of *I*-section model contains 27 DOF. The first stability modes of the tapered column are of a local shape.

Table 2. The results of the variable section solutions

FE model	Characteristics		
	FE	DOF	λ_{cr}
Shell FE	1536	9600	1,00
SAFE	32	864	0,94
	16	432	0,93
	8	216	0,91

The numerical results show the good agreement between the proposed model and the results obtained from the classical theory and solutions by a shell FE model (Table 2). Therefore, the proposed models are actual in the range of thin cross-sectional walls and fill the gap between the theory of thin-walled beams and shells.

6. Conclusions

On the basis of the investigation results the following main conclusions have been made:

- The proposed computational tools for the processor allow assembling of the global linear and geometric stiffness matrices by formation and assembling of individual SE and present the global matrices in the form as they are used in conventional FE software;
- Application of symbolic manipulations and computer algebra allowed to develop a software tool for derivation of explicit expressions of characteristic matrices of SE in the form of FE library subroutines;
- The proposed software tools for pre- and post-processors enable to perform communication between a user and conventional FE software using definitions and physical interpretation of the SAFEM;
- Comparison of numerical results obtained by using the SAFE and conventional shell FE analysis proves the efficiency of the proposed SAFE and developed software tools as well as suitability of them for stability analysis of thin-walled beams.

References

[1] ANSYS, User's Manual. *SAS IP Inc.*, 2003.

[2] **J. Argyris, P.C. Dunne.** The General Theory of Cylindrical and Conical Tubes under Torsion and Bending Loads; Single- and Many-Cell Tubes of Arbitrary Cross-Section etc. *Journal of the Royal Aeronautic Science, Vol.* 51, 1947, 199-269, 757-784, 884-930; *Vol.* 53, 1949, 461-483, 558-620.

[3] **J. Argyris, R. Kačianauskas.** Semi-Analytical Finite Elements in the Higher-Order Theory of Beams. *Computational Methods in Applied Mechanics and Engineering, Vol.* 138, 1996, 19-72.

[4] **R. Barauskas.** Baigtinių elementų metodo pagrindai. *Technologija, Kaunas*, 1998.

[5] **G.M. Barsan, C.G. Chiorean.** Computer Program for Large Deflection Elasto-Plastic Analysis of Semi-Rigid Steel Frameworks. *Computers and Structures, Vol.* 2, 1999, 699-711.

[6] **R.S. Barsoum, K.H. Gallagher.** Finite Element Analysis of Torsional and Torsional-Flexural Stability Problems. *International Journal for Numerical Methods in Engineering, Vol.* 2, 1970, 335-352.

[7] **K.-J. Bathe, N.-S. Lee, M.L. Bucalem.** On the Use of Hierarchical Models in Engineering Analysis. *Computer Methods in Applied Mechanics and Engineering, Vol.* 82, 1990, 5-26.

[8] **K.-J. Bathe, E. Wilson.** Numerical Methods in Finite Element Analysis. *Printice-Hall, New Jersey*, 1976.

[9] **R. Belevičius.** Computer Algebra in Finite Element Method. *Technika, Vilnius*, 1994.

[10] **R. Belevičius, R. Kulvietienė, G. Kulvietis.** Generating Matrices of Finite Elements Using Computer Algebra System VIBRAN. *Proceeding of the 5th Nordic seminar on Computational Mechanics, Aalborg, Denmark*, 1992, 78-81.

[11] **R. Belevičius, P. Pedersen.** Analysis and Sensitivity Analysis by Computer Algebra for the Third Order Plate Finite Element. *Computers and Structures, Vol.* 49, 1993, 243-252.

[12] CALFEM: A Finite Element Toolbox to Matlab. Version 3.3. *Lund University*, 1999.

[13] **J. Cheng, X. Ru-Cheng, H.-F. Xiang, J.-J. Jiang.** NASAB: a Finite Element Software for the Nonlinear Aerostatic Stability Analysis of Cable-Supported Bridges. *Advances in Engineering Software, Vol.* 34, 2003, 287-296.

[14] Compaq Fortran, Language Reference Manual. *Compaq Computer Corporation*, 1999.

[15] **M. Grinsberg, G. Clifford, F. Felker, L. Komzsik, S. Yunus.** Future Directions for Commercial Finite Element Software. *Advances in Engineering Software, Vol.* 29, 1998, 487-492.

[16] **N.I. Ioakimidis.** Application of Mathematica to the Direct Solution of Torsion Problems by the Energy Method. *Computers and Structures, Vol.* 43, 1992, 803-807.

[17] **J. Jonsson.** Determination of Shear Stress, Warping Functions and Section Properties of Thin-Walled Beams Using Finite Elements. *Computers and Structures, Vol.* 68, 1998, 393-410.

[18] **R. Kačianauskas.** Computer Methods in Multilevel Modelling of Beams and Shells. *Technika, Vilnius*, 1995.

[19] **R. Kačianauskas, M. Samofalov.** Semi-Analytical Finite Elements for the Stability Analysis of Thin-Walled Beams. *Proceedings of the 2nd European*

Conference on Computational Mechanics, Cracow, 2001, CD-ROM: 30.2.1-20.

[20] **E.-D. Kim, S.-K. Park, K.-I. Lim.** A Prototype for Step-based Structural Design and Analysis of Building Structural Frames. *European Congress on Computational Methods in Applied Sciences and Engineering ECCOMAS, Barcelona*, 2000, CD-ROM: 6.3.46-62.

[21] **M.Y. Kim, S.P. Chang, S.B. Kim.** Spatial Stability Analysis of Thin-Walled Space Frames. *International Journal for Numerical Methods in Engineering, Vol.* 39, 1996, 499-525.

[22] MATHEMATICA, User's Guide. *Wolfram Research Inc.*, 2003.

[23] **S. Moaveni.** Finite Element Analysis. Theory and Application with ANSYS. *Printice-Hall, New Jersey*, 1999.

[24] MSC.Nastran. Reference Manual. *MSC.Software Inc.*, 2003.

[25] **J. Nagabhushanam, C.J. Srinivas, G.H. Gaonkar.** Symbolic Generation of Elemental Matrices for Finite Element Analysis. *Computers and Structures, Vol*. 42(3), 1992, 375-380.

[26] **K.A. Noor.** Survey of Some Finite Element Software Systems. Finite Element Handbook, edited by H. Kardestuncer, D.H. Norrie. *McGraw-Hill, New York*, 1987, 4.259-4.288.

[27] **J.P. Papangelis, G.J. Hancock.** Computer Analysis of Thin-Walled Structural Members. *Computers and Structures, Vol.* 56(1), 1995, 157-176.

[28] **B.N. Parlett.** The Symmetric Eigenvalue Problem. *Printice-Hall, New Jersey*, 1980.

[29] **A. Prokič.** Computer Program for Determination of Geometrical Properties of Thin-Walled Beams with Open-Closed Sections. *Computers and Structures, Vol*. 74, 2000, 705-715.

[30] **M. Samofalov, R. Kačianauskas.** Application of semi–analytical finite elements for thin–walled beams with distortion. *Proceedings of the 14th Nordic Seminar on Computational Mechanics NSCM–14, Lund*, 2001, 79–82.

[31] **M. Samofalov.** Semi-Analytical Finite Elements for Thin-Walled Beams and Their Application to Stability Problems. *PhD Thesis. Vilnius Gediminas Technical University, Vilnius*, 2002.

[32] **M.E. Teodorescu, V. Banut.** A Comuter Program for Large Displacements of Space-Frames Structures. *Proceedings of the 2nd European Conference on Computational Mechanics, Cracow,* 2001, CD-ROM, 15.6.1-14.

[33] **V.Z. Vlasov.** Thin-Walled Elastic Beams. *Izrael Program of Scientific Translation, Jerusalem*, 1961.

[34] **E.L. Wilson, M. Hoigt.** A Computer Adaptive Language for the Development of Structural Analysis Programs, *Computers and Structures, Vol.* 19, 1984, 321-328.

[35] **O.C. Zienkiewicz.** The Finite Element Method in Engineering Science. *McGraw-Hill, London*, 1971.

[36] **O.C. Zienkiewicz.** The Finite Element Method. *McGraw-Hill, London*, 2000.