Experience of analysis of bridge structures

Alexey Kniga, Igor Kolyushev, Alexandre Kountsevich, Dmitry Maslov, Maxim Pashkovsky* and Vladimir Slivker
JSC "Institute Giprostroymost-Saint-Petersburg",
Yablochkova 7A, 197198, Saint Petersburg, Russia
e-mail: slivker@gpsm.ru

Abstract

This report gives some results of the most interesting and unusual, in terms of structure stress, bridge analysis, designed at Institute Giprostroymost – St.Petersburg. The commercial software for technical application SCAD (Ukraine) and GTSTRUL (USA) as well as Giprostroymost software, and improved available programs (push-launching of bridge span, converter for improvement of the input language data GTSTRUDL etc.) serves as the analysis toolkit. The features of mathematical modeling and the technique of analysis for three bridges are discussed as demonstration examples.

Keywords: design models, analysis of bridges

1. Introduction

Modern engineering software gives enormous variety for civil engineers. In the long list of up-to-date commercial software there is a way to get one, which helps to analyze particular construction. Here are the most popular versions of such types of software: GTSTRUDL (USA), SCAD (Ukraine), MicroFE (Russia), Lira (Ukraine), ANSYS (USA), Robot (French), Strauss (Australia), NASTRAN (USA), ABAQUS (USA), STAAD (USA), COSMOS (USA), LUSAS (UK), DIANA (Netherlands), RM2000 (Austria), ADINA (USA), etc. It looks like that the sphere of application is almost the same for any software, but not completely.

Tough competition between software developers for civil engineering inspires marketing companies for aggressive publicity by emphasizing the advantages of their products. As can be seen by means of these commercials, on purpose or by chance (looks like on purpose) you have a feeling of unbelievable importance of engineering software features and useless of engineer's intelligence. What is it, for any brainless "mouth potato" work?!

We consider it as a wrong attitude to minimize the qualification importance of user.

Usually software technical documentation contains a few words, or even nothing, for creation of analytical models. But this is a keystone for using high-level knowledge and professionalism. Sometimes users are so creative that they do enhance the sphere of implementation of engineering software his own unique way. In present technical literature, this aspect of engineering software implementation is poorly described.

A lot of worthy to use creation of analytical models and methods of calculations for particular software orally gives from one user to another. That is why we want to focus on Russian book [1], with detailed description of different types of useful customized mechanical models. This book will be issued this year by Springer Verlag [2] in famous Foundations of Engineering Mechanics Series.

Special analytical models and cost effective options usually appeared due to practical demands, especially for bridge construction.

Here are some examples of mechanical models originally developed by authors of present report.

Those examples can be used as a sample guideline for similar engineering tasks.

The second point of this article is to demonstrate the shortcoming of modern engineering software and to emphasize that user forced to develop his own customized software. The purpose of that software is, definitely, not to exchange the universal program, but to improve and enhance it with proper auxiliary features.

We want to share an experience of JSC "Institute Giprostroymost-Saint-Petersburg", which may be useful for civil engineers.

2. The viaduct over Moscow-Kiev railway at the Moscow ring road

This viaduct was constructed in 1999. It consists of continues four span girder 55+80+80+55m. The task of construction was not to block, even for short time the railroad traffic, so engineers were forced to perform push-launching method (equipped with 70ml launching nose).

Concerning of the bridge analysis there were following problems:

- The angle of crossing of the viaduct and railroad was 60º, so those piers had to be settled at the same angle to longitudinal deck axis. Such geometry results deck torsion.
- Cross-section of the deck is a twin box-girder with steel plate on the top bonded with braces (Fig. 1). At the push-launching time some parts of the deck have no upper flange) and work as a "thin walled member" (Fig. 2). Shear center of given structure, considerably displaced toward vertical direction of gravity center. Relatively small rigidity of structure torsion, force to pay attention at warping phenomena.
- During push-launching process, the deck could be separated from rollers. This mechanical system is an example of one with unilateral constraint.

The problem of calculations for push-launching process consists of setting the top level of pier rollers and evaluation of box girder stresses. At first sight, it looks like it is a simple

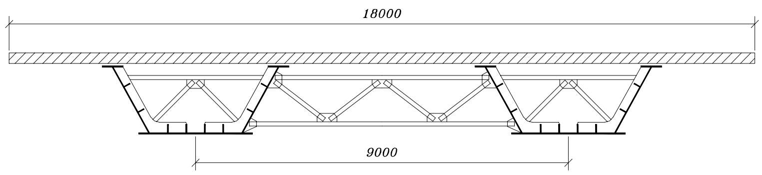

Figure 1: Cross section of the viaduct girder

model you can apply by dividing whole span on many finite elements, having plane stress and bending stress.

Above mentioned model can be applied with neglecting warping phenomena during deck torsion, cause you can watch it on finite element model. Ribs number forms sophisticated geometry of box girder cross-section, so you should add a lot of finite elements and nodes to discrete analytical model. It is clear, huge number of degrees of freedom is not an obstacle for modern methods of calculation, but such an extreme approach to it has number of shortcomings:

- Wasting time and resources for preparation and verification of input data of discrete system;
- Huge amount of output data to analyze;
- Bad conditions for governing equations, results bigger chances for errors.

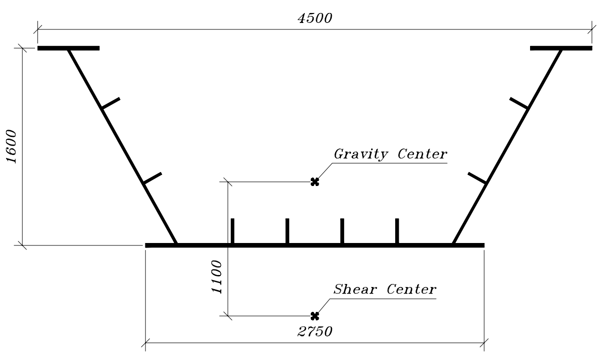

Figure 2: Cross section of separate beam

At the same time, a simple mechanical model of girder which allows us easy to define basic features of stress-strain state. Definitely, we talk about "member model" of girder, not a beam in classic point of view, as a thin-walled member with open cross-section. Evidently, the basic Vlasov's theory of thin-walled member is rather suitable for real bridge structure. But here is another problem to come – present commercial software do not equipped with the 7-th degree of freedom (warping), which arises by applying V.Z. Vlasov's theory. But it is not a big deal and here is the decision.

For applying given civil engineer software for construction analysis to the stage of to push-launching, a special mechanical model was developed, called "Bi-member model". This one allows us to implement normal software with no modification.

The idea of this model is to exchange original thin-walled member on two members: one (Fig. 3) F – "false member" is responsible for accumulation of energy of warping, other parts of energy belongs to M – "main member" on Fig. 3. Both systems work similarity (Bi-member model and thin walled one) insured by applying constraints on system nodes movements. Those constrains prevent the nodes movement of false member along X_F axis and along one of the main inertia axis. Displacements of false member nodes along another main axis connected with twisting angle θ_x of main member by linear function

$$w_F = r\theta_x, \tag{1}$$

where, w_F – denotes as a gravity center of false member displacement toward Z_F axis; r – a kind of constant, a user can apply at his own discretion. At given circumstances, the seventh degree of freedom of thin-walled member nodes converts to normal degree of freedom θ_{yF} of false member because

$$\theta'_x = w'_F / r = -\theta_{yF} / r. \tag{2}$$

This circumstance allows us to apply present software (with no upgrades) for the analysis of thin walled member.

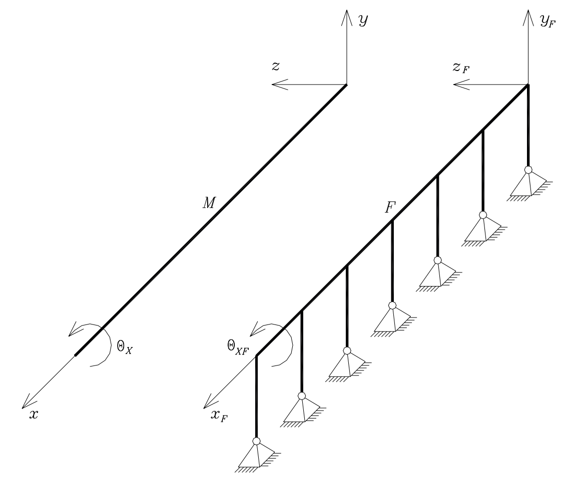

Figure 3: Bi-member model

The report [3] presents detailed description of "Bi-member model" plus mathematical background with implementation of required formulas for assigning of basic data to false member and main member in accordance with geometrical data.

Application of "Bi-member model" to the analysis of construction helped to fulfill required engineering calculations in time with precise estimation of the construction stress-strain state.

The results of calculations showed significant influence of restrain warping on structure stress–strain state.

Bitorque stress, in the most dangerous parts of thin walled member, reached almost 15% from normal bending stress.

The results comparison of geodetic shooting, made during push-launching process, with numerical forecast showed relatively high precision of Bi-member model (total deviation of displacements did not exceed 5%).

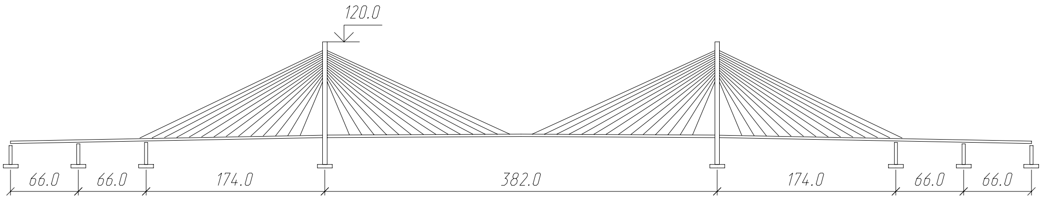

Figure 4: Cable-stayed bridge over tne Neva river – St.Petersburg, Russia

3. The cable-stayed bridge over Neva River in Saint-Petersburg

The drawing of this bridge is shown on Fig. 4. The bridge is in the stage of construction. There are special circumstances for the analysis of construction:

- Geometrical nonlinearity, coarse of stay-cables sag and high level of pylon and deck stress.
- Bridge structure dynamic wind analysis.
- Application of fragmentation technique for the analysis of stress-strain state of particular joints of steel pylon and deck.

Concerning geometrical nonlinearity, due to stay-cables sag, it is not a serious obstacle being used in developing mechanical model. The special element ensured in software (for example in GTSTRUDL, SCAD, LUSAS etc.) called CABLE-element. But this element suits only to static analysis of the structure. At the same time, for cable-stayed bridges dynamic analysis should be conducted, especially for mode shapes and natural frequencies definition of the bridge structure to the construction period (construction of a single pylon, cantilever scheme of a span, completed structure). There is no way to model dynamic characteristics of the bridge without dynamic analysis.

Let us mark special model of cable for the purpose of dynamic analysis. As we focus on small vibrations within the equilibrium state of the structure, the response of cable-stayed system can be described by implementation of *linearized model*. This model holds stress state of cable-stayed system unchanged at small vibrations.

If engineer has the software (with instant stiffness matrix of cable included) there is no problem. But if not, there is a way for engineer to solve the task by preparing tangent stiffness matrix of cable by means of input data.

Axial displacements of cables' nodes are not hard to analyze by applying well-known Ernest formula. This formula gives us, so-called, *effective cross-section area* of a rod A_0, which equals to instant axial stiffness of cable

$$A_0 = A\frac{1}{1+\gamma^2 l^2 E/(12\sigma^3)}, \quad (3)$$

where A – cable cross-section, E – Young module, l – horizontal projection of a strand, γ – density of strand's material, σ – cross-section cable stress.

But classic truss element does not suit for transversal displacements, because there is no way to create components of the tangent stiffness matrix (transversal displacements does not produce required reactions).

We can develop the model for transversal displacements by use of bending rod model. As a result each cable exchanged on classic beam with linearized stiffness. Here are three following models fig. 5

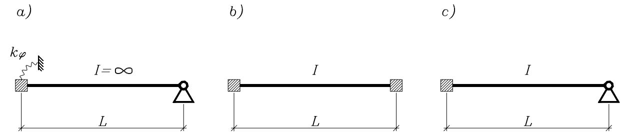

Figure 5: Linearized cable elements

Stiffness characteristics of these elements can be assigned in accordance with following formulas

$$a)\ k_\varphi = NL \quad b)\ I = \frac{NL^2}{12E} \quad c)\ I = \frac{NL^2}{3E} \quad (4)$$

where k_φ – model spring stiffness in model *a*; I – moment of inertia of a beam in models *b* and *c*; E – Young module of the beam; N – axial force of cable element. Sure, the nodes of models *b* and *c* should be fixed from rotation.

Description and formal proving of those models are given at [1,2]. The models are applicable to any commercial software.

We want to emphasize, that implementation of linearized cable elements to dynamic model of cable-stated bridge, showed small influence of transversal reactions on given dynamic characteristics (frequencies and modes). Such an influence makes sense only for transversal vibrations of the girder, especially for cantilever stage.

But above mentioned models are very essential for suspensions structures. Suspension bridge over Nevelskoy Strait is an example of such a structure, analyzed by authors of this report in JSC "Institute Giprostroymost-Saint-Petersburg". In case of suspension bridge, negligence to transversal reactions leads to rough mistakes, it results structural instability.

Analysis of structural stability required separate modeling. The dangerous errors were found in software. We came to conclusion, that almost totally commercial software are able to make mistakes while defining conditions of structural instability especially in cases of implementation perfectly rigid elements.

Let us take a look and analyze on published examples [4] of that widespread errors, that happens by using of software. As for now, the only one well known to authors commercial software is SCAD (Ukraine), where those errors are corrected. We recommend this software for the tasks of stability analysis with no risk of errors.

One more model, called *compressed-bend element* worthy to apply, was implemented to the analysis of cable-stayed bridge in Saint-Petersburg. Definitely, talking about characteristics of Neva bridge, (longitudinal bend of pylon legs and girder) those effects had insignificant values. But the idea of such model may be useful in another situations. The model of *linearized compressed element* consists of common beam supported by springs as it shown on the Fig. 6.

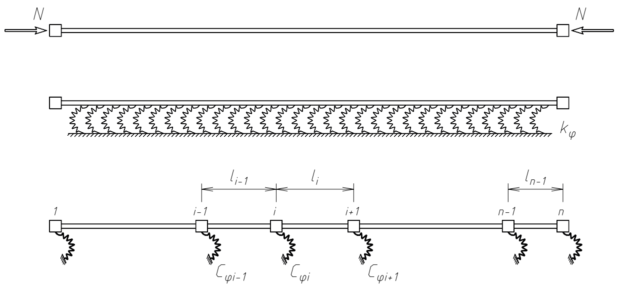

Figure 6: Linearized compressed element

Spring stiffness C_φ in *i*-node, on Fig. 6, should be assigned as follows

$$C_\varphi = -N(l_{i-1} + l_i)/2, \tag{5}$$

where N is axial force of the beam (positive while compressed); l_i – length of *i*-segment of beam. We have to mention, some commercial software do not allow to use negative stiffness characteristics of any elements. Within restrictions like these, it is hard for user to apply described models or even impossible to do it. It is a disadvantage only, from user's point of view.

Finally, let us look at fragmentation technique, widespread in structure analysis. This technique, for example, was implemented for Neva bridge pylons and deck joints analysis. Actually, the idea of fragmentation, as an individual technique for static analysis, is based on famous Saint-Venant principle.

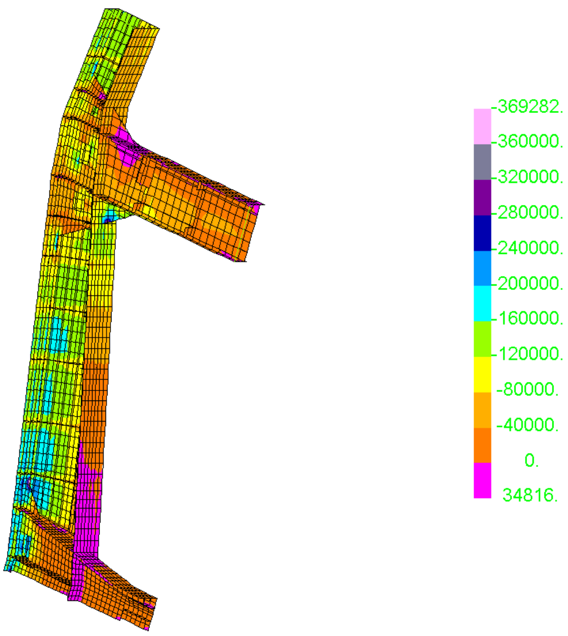

Figure 7: Fragment of Neva bridge pylon

Fig. 7 shows an example of Neva pylon fragment with finite elements net and results of analysis, calculated by means of GTSTRUDL. It is clear that we can apply fragmentation technique with any software based on finite element method. But it would be useful, if those commercial software give a possibility to user to apply fragmentation technique at his own disposal, to any given structure node with automatic definition of forces acting at highlighted fragment.

Such type of analysis technology called *fragmentation technique*, being provided in SCAD.

One more typical aspect, we should mention concerning Cable-Stayed bridges is a role of multi-stages of assembling. For this purpose we need to find stress-strain state of the bridge at different stages of deck construction and strands installation. This type of analysis is very sophisticated and requires high precision of calculations. That is because at the stage of assembling the most important things to control are geometrical characteristics, such as global coordinates of certain points of the deck.

Traditional way of cable-stayed bridges analysis for the mounting stages of structure known as *backward analysis*. That means mental backward dismantling of the bridge. Also we can apply step-by-step analysis called "*forward analysis* " with total sum increments of stress and displacement components corresponding to any stage of construction. Mostly, *forward analysis* is being applied additionally after *backward analysis,* as a method of assessment control or cables installation verification. But, the possibility of application *backward analysis* assumes, that engineer knows beforehand stress-strain state of the bridge in service.

Let us call, for convenience, geometrical position of a bridge in service *deformed configuration of structure.* That is why, by means of *backward analysis*, starting from *deformed configuration of structure* as an primary one, engineer comes to the initial (undeformed) configuration of mechanical system. But the same configuration could be found without step-by-step analysis. This way we can get complete geometrical information about every single unit (element) of construction. By means of all geometrical "*preps*" we can get stress-strain state of *deformed configuration of structure* any time at any stage of construction without step-by-step analysis.

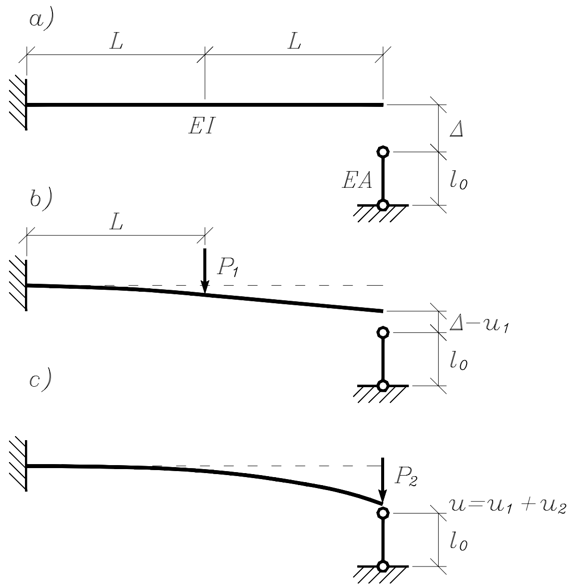

Figure 8:

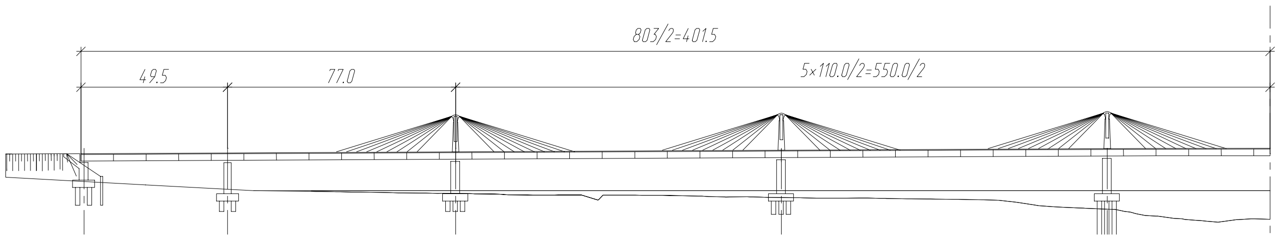

Figure 9: Extradosed Composite Bridge over tne Daugava river – Riga, Latvia (half of the bridge)

This statement looks clear, because we deal with conservative systems – it means independence of stress-strain state from loading sequence.

Let us show it on simple example. On Fig. 8-*a* we can see undeformed configuration of mechanical system. This system consists of cantilever beam with a length of $2L$ and vertical truss rod with a length of l_0 with axial stiffness EA. We guess that clearance (even negative one) between the nock of cantilever beam and truss equals Δ. Let us apply force P_1 to the middle of the beam at the primary stage of assembling (Fig. 8-*b*). At the second stage of assembling the beam is connected to the truss (Fig. 8-*c*) and additional force P_2 is being implemented to the nock of the cantilever beam. It is absolutely clear, that stress-strain state of mechanical system can be analyzed instantly after second stage of assembling without preliminary analysis of the primary assembling stage. For this purpose we should simultaneously apply forces P_1 and P_2 and dislocation Δ to the system. It is easy to notice that this statement still matters in cases of non linear (but conservative!) mechanical systems. For example, links between vertical truss rod's shortening and internal force of that rod could be non-linear.

Application of step-by-step procedure, having in disposal kinematics clearances to underformed system, is a waste of engineer's time. That is why, for Neva bridge analysis above mentioned technique at each assembling stage was implemented.

4. The South Bridge over Daugava River in Riga, Latvia

The bridge over the Daugava river represents multispan structure 49.5+77+5×110+77+49.5m (Fig. 9) well known Extradosed system with 6 traffic lanes. The project of this bridge is still in developing stage. Low pylons (11.3m height) are settled on the deck. Each pylon has 8 pairs of cables.

Due to number of stages of deck concreting and double stage strands stressing so-called *genetic nonlinearity* takes place. This term was recently introduced (see [2,3]). It denotes the accumulation law of stress-strain state, while transferring from one stage to another.

It is important, that geometrical preps are unknown beforehand. If you add new element to mechanical system on (k+1)–stage of assemblage, you may define geometrical characteristics of that element in deformed configuration of mechanical system on k–stage of assembling.

If structure works as a physical linear and geometrical linear one, the number of present software are able to consider genetic nonlinearity. That software could varies concerning convenience of application. For instance, in GTSTRUDL you can initialize an "activation" or "deactivation" mode for selected elements or parts of structure. After that you should indicate loading combination only. Similar mode named "montage" is presented in SCAD.

The analysis of bridge over Daugava river was directed for searching rational sequences of strands installation and a sequence of deck concreting.

5. Programs developed by JSC "Institute Giprostroymost-Saint-Petersburg"

Earlier we noticed the benefit of "Bi-member model" for the purpose of bridge design. At the same time, we should mention that analysis of thin-walled members requires assessment of additional member geometrical properties expanding the typical ones. These values are shear center location, sectorial moment of inertia, sectorial static moments, and sectional areas.

At the time of viaduct analysis (over Kiev railroad) we had not software to calculate those properties, GTSTRUDL does not equipped with that procedure. Because of that obstacle, those properties were calculated manually, which took a lot of time with possibility of errors due to immense amount of calculations. So, we decided to develop our own program. This program is able to calculate all required geometrical characteristics for different types of cross-sections: opened and closed thin-walled and solid. The program developed by JSC "Institute Giprostroymost – St.Petersburg" bears the name "GeomyX". An example of application of GeomyX for above mentioned viaduct deck cross-section showed on Fig. 10.

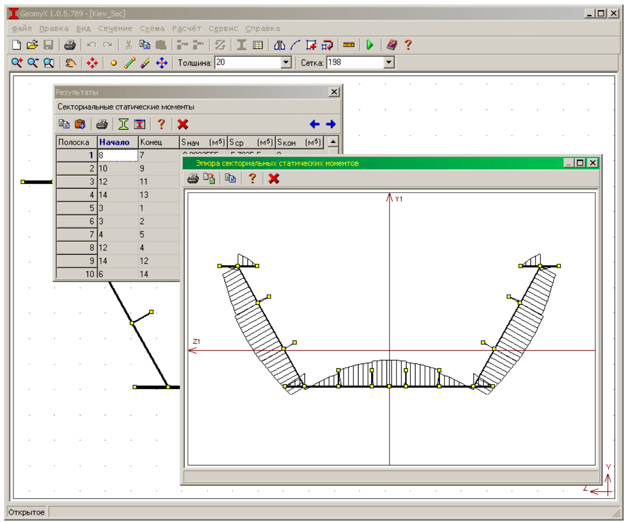

Figure 10: "GeomyX" screenshot

This program allows to input cross-section data both in table and graphical modes, or import data from AutoCAD. Each value describing a cross-section may be dependent on user specified parameters or other values. Data may be inputted starting with an empty section or a template. The program provides catalogues of templates and profiles which may be expanded by user. The output data of GeomyX might be exported to other applications or processed within the program itself.

Now there are some software packages supplied with applications to calculate geometrical properties of thin-walled cross-sections (for instance, SCAD or ROBOT). At the same time, there are number of programs not provided by this feature. In these cases "GeomyX" seems to be useful for an engineer.

The second program we would like to introduce is "ExpConv" (Expressions Converter). This is a preprocessing unit that we use to expand the problem oriented language of GTSTRUDL. As we suppose, the widespread interactive graphical input mode may not completely substitute the text command mode, particularly for bridge structures, which stiffness properties often vary from one element to another. As for GTSTRUDL, this software supplied with command language to input data. Our experience shows that many values of mechanical model (such as coordinates, topology, stiffness, and loadings) can be described parametrically.

We made a new mechanical model each time we changed parameters. It caused immense calculations based on a few formulas. These operations took a lot of time and generated errors. The "ExpConv" program helps to solve these problems. This program reads a text consisting of GTSTRUDL commands with expanding statements and produces new text that contains GTSTRUDL commands only. Let us notice that the "ExpConv" is not binded to GTSTRUDL and may be used in many different applications.

We are not able to introduce the detailed description of ExpConv and statements of its language here. But we do want to emphasize the convenience and benefits of this program. We can say that since our company engineers started to use this converter, they do not want to perform everyday engineering task calculations without this program.

We would like to appeal to the commercial software developers: having advanced language tools in addition to graphic interactive ones makes the issued product more attractive.

References

[1] Perelmuter, A.V. and Slivker, V.I., *The analysis of constructions – models and interpretations*, 2nd edition, "Steel" Publ. Co, Kiev, 2002, (in Russian)

[2] Perelmuter, A.V. and Slivker, V.I., *Numerical Structural Analysis. Models, Methods and Pitfalls.* Springer Verlag, 2003, (to be published).

[3] Kountsevitch, A.O. and Slivker, V.I., Bi-member model of thin-walled member of open cross section, *ECCCM-2001. 2nd European Conference on Computational Mechanics. Solid, Structures and Coupled Problems in Engineering.* Cracow, Poland, June 26-29, 2001. Abstracts, Vol.2, pp.1048-1049. Kraków, Vesalius, 2001. (Full paper on enclosed CD-ROM).

[4] Perelmuter, A.V. and Slivker, V.I., On an error of a mysterious nature that happenes in software when analysing mechanical systems for buckling, *NSCM-15. Proceedings of the 15th Nordic Seminar on Computational Mechanics*, Lund E., Olhoff N. and Stegmann J. Eds, Aalborg University, Denmark, pp.229-232, 2002