10.1515/tvsb-2015-0004

Transactions of the VŠB – Technical University of Ostrava

Civil Engineering Series, No. 2, Vol. 15, 2015

paper #04

Bence BALOGH[1], János LÓGÓ[2]

OPTIMIZATION OF CURVED PLATED STRUCTURES WITH THE FINITE STRIP AND FINITE ELEMENT METHODS

Abstract

The aim of this study is to compare two available numerical tools for solving of partial differential equations for the optimal design of structures. In the past years numerous methods were developed for topology optimization, from these we have adopted the optimality criteria (OC) approach. The main idea is that we state the optimal conditions, that the minimizer has to fulfil at the end of an iterative proves. This method however is not a general one, only advantageous in the case of separable problems, but comes with fast speed, easy programming, and a relative insensitivity of computational time to the number of variables. In the paper we suggest a new method for the elimination of a numerical error, the so called ‘checkerboard pattern’. In the presented examples we applied one loading case and an elastic material behaviour. The cost function is the net weight of the structure and upper bound of the compliance is set as the optimality constraint.

Keywords

Topology, optimization, optimality criteria, finite element method, finite strip method, FEM, FSM.

1 INTRODUCTION

The definition of a structure is defined by J.E. Gordon as “any assemblage of materials which is intended to sustain loads.” The aim of topology optimization is to find the structure, which fulfils these criteria in the best possible way.

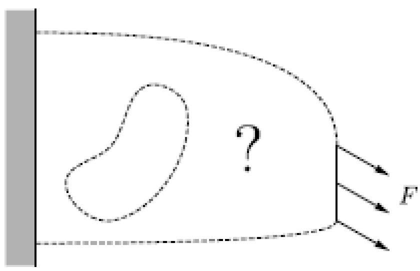

Fig. 1: The programming problem

---

[1] Bence Balogh, PhD student, Budapest University of Technology and Economics, Műegyetem rkp. 3, Budapest, Hungary, e-mail: bbence@eik.bme.hu.

[2] János Lógó, Associate professor, Budapest University of Technology and Economics, Műegyetem rkp. 3, Budapest, Hungary, e-mail: logo@ep-mech.me.bme.hu.

For this to happen, in the first place we have to define what we mean by being the “best”. Assuming a correlation with economic design, we consider a structure to be optimal, if it produces the lowest net weight. The function which is to be minimized is called as an objective function. This function assigns a scalar to every design of a structure, where a smaller value means better, hence the task is to find the minimizer of it. For the problem to be well defined, we need to state restrictions, which exclude the trivial solution. It is commonly used to achieve this by setting an upper bound on the compliance of the structure. This is no other as the external work of the static loads, which in the case of a linearly elastic material equals the double of the strain energy. It is foreseeable by multiplying the statical state equation of a FEM model by the vector of nodal unknowns $\{u\}^T$:

$$\{u\}^T[K]\{u\} = \{u\}^T\{f\}. \tag{1}$$

In this equation $\{u\}$ and $\{f\}$ denotes the vector of nodal unknowns and forces, respectively, $[K]$ is the global stiffness matrix of the system. From now on, the expression on the left will be denoted with C. The design is described with the design variables, which usually are some thickness, or other geometrical property of the structure. A state variable is a function or vector expression, representing the response of the structure to the given load. Further restrictions can be stated to the design or the state variables. In the scope of the present paper, the discussed problems could be mathematically stated in the following form:

$$\begin{cases} \min\limits_{\{u\}} V(\{x\}) \\ subject\ to \quad \{u\}^T[K]\{u\} < C_0, \\ and \quad [K]\{u\} = \{f\}; \end{cases} \tag{2}$$

where $V(\{x\})$ is the objectie function of the design variables $\{x\}$, C_0 is the mentioned upper bound on the compliance. The last equation is the state equation of the structure, which, according to the previously mentioned, equals to a constraint on the state variables.

The stated problem is a constrained programming one. This can be transformed to an unconstrained one using penalty functions, or more often the method of Lagrange multipliers. The latter one can only be used, if the constraints are written in the form of equality. With the expanded version of this method, one can state the general criteria, which the design variables have to satisfy in the minimum point of the cost function these are called as the Karush-Kuhn-Tucker (KKT) conditions. There are several methods to solve the programming problem and find the minimizer, among those, which solve the KKT conditions directly, are referred as optimality criteria methods [1][2][3].

In the next chapters we will investigate the possibilities in this method, in the meaning of the applied numerical tool for stress analysis. Since the two major numerical techniques for solving partial differential equations (PDEs) in structural engineering is the finite element method (FEM) and the finite strip method (FSM), these will be the subject of comparison. We adopt the assumptions of the Mindlin-Reissner plate theory.

It is also important to highlight the question of convexity. A general programming problem is said to be convex, if the both the cost function and the functions describing the constraints are convex. This concept comes with the favourable property that the local and global minimizer coincides. The question of convexity, however, is a complicated task, but has proven for the case of (2). The details of the proof can be found in the literature [4].

In the forthcoming chapters we assume a solid understanding of the introductory concepts, and the two numerical tools, as mentioned above.

2 THE FINITE ELEMENT AND THE FINITE STRIP METHOD

Due to being widespread in many field of engineering, we don't introduce the theoretical aspects of the FEM, but focusing instead on the FSM.

Although the most successful and current numerical tool of the past decades for solving engineering problems is the FEM, we share the view that one has to take every opportunity, which allows the theoretical ideas to take place. This gives rise to the application of FSM, as a semi-analytic modification of the FEM. The foundations of the theory were laid down by Cheung in 1960 by combining the usual finitization technique with analytic results. He realised, that in certain cases, we can exploit the specialties of the geometry, which allows us to make some simplifications. A few examples of structures with such beneficial geometrical properties are illustrated on Figure 2.

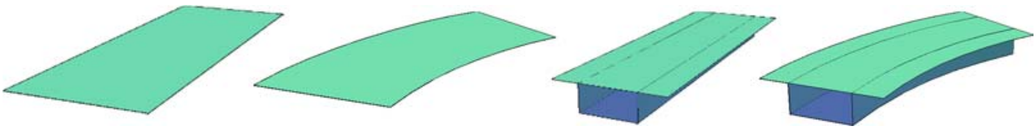

Fig. 2: Structures with beneficial geometrical properties for the FSM

So, the main difference lies in the way of finitization. The two different approaches are illustrated on Figure 3.

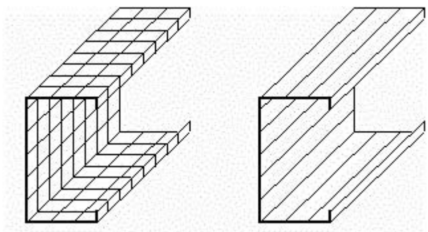

Fig. 3: Finitization of the geometry with FEM (left) and FSM (right)

The big advantage of FSM is that if the cross section of the structure is constant, the number of unknowns is insensitive to the span length. This property is also fruitful, when creating a mesh generator, since the discretization happens in lower dimension, even in the case of a curved axis.

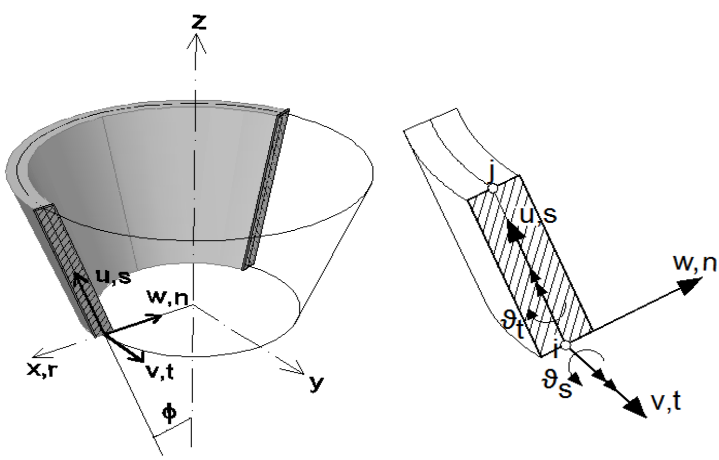

Fig. 4: Sign convention for displacements in a tronco-conical shell

For a more general discussion, the expressions are formulated in a tronco-conical coordinate system, shown in Figure 4. Then, the simpler geometries can be deduced as a particular case of this formulation.

The displacement field at any point of the shell can be expressed as:

$$\begin{aligned} u(s,\theta,n) &= u_0(s,\theta) + n\vartheta_s(s,\theta), \\ v(s,\theta,n) &= v_0(s,\theta) + n\vartheta_t(s,\theta), \\ w(s,\theta,n) &\cong w_0(s,\theta), \end{aligned} \tag{3}$$

where u, v and w are the displacements of a typical point in the s, t and n directions, while ϑ_s and ϑ_t are the normal rotations contained in the planes sn and tn. The elements of the small strain tensor are:

$$\begin{aligned} \varepsilon_s &= \frac{\partial u}{\partial s}, \\ \varepsilon_t &= \frac{1}{r}\frac{\partial v}{\partial \vartheta} + \frac{u}{r}\sin\phi - \frac{w}{R_t}, \\ \gamma_{st} &= \frac{1}{r}\frac{\partial u}{\partial \vartheta} + \frac{\partial v}{\partial s} - \frac{v}{r}\sin\phi - \frac{n}{R_t}\frac{\partial v}{\partial s}, \\ \gamma_{sn} &= \vartheta_s + \frac{\partial w}{\partial s}, \\ \gamma_{tn} &= \vartheta_t + \frac{1}{r}\frac{\partial w}{\partial \vartheta} + \frac{v}{R_t}. \end{aligned} \tag{4}$$

The interpolation model for the displacement field has to be choosen so, that the trigonometric terms must satisfy the prescribed boundary conditions. In our case the investigation is restricted to the case of the classical FSM, where simply supported boundary conditions are adopted at the two ends of a strip element. Therefore:

$$\begin{aligned} u(s,\theta) &= \textstyle\sum_{l=1}^{n} {u_0}^l(s)\cdot\sin\left(\frac{l\pi}{\alpha}\theta\right) = \sum_{l=1}^{n}\sum_{i=1}^{n_e} N_i(s)\cdot {u_0}_i^{\,l}\cdot\sin\left(\frac{l\pi}{\alpha}\theta\right), \\ v(s,\theta) &= \textstyle\sum_{l=1}^{n} {v_0}^l(s)\cdot\sin\left(\frac{l\pi}{\alpha}\theta\right) = \sum_{l=1}^{n}\sum_{i=1}^{n_e} N_i(s)\cdot {v_0}_i^{\,l}\cdot\sin\left(\frac{l\pi}{\alpha}\theta\right), \\ w(s,\theta) &= \textstyle\sum_{l=1}^{n} {w_0}^l(s)\cdot\sin\left(\frac{l\pi}{\alpha}\theta\right) = \sum_{l=1}^{n}\sum_{i=1}^{n_e} N_i(s)\cdot {w_0}_i^{\,l}\cdot\sin\left(\frac{l\pi}{\alpha}\theta\right), \\ \vartheta_s(s,\theta) &= \textstyle\sum_{l=1}^{n} {\vartheta_s}^l(s)\cdot\sin\left(\frac{l\pi}{\alpha}\theta\right) = \sum_{l=1}^{n}\sum_{i=1}^{n_e} N_i(s)\cdot {\vartheta_s}_i^{\,l}\cdot\sin\left(\frac{l\pi}{\alpha}\theta\right), \\ \vartheta_t(s,\theta) &= \textstyle\sum_{l=1}^{n} {\vartheta_t}^l(s)\cdot\cos\left(\frac{l\pi}{\alpha}\theta\right) = \sum_{l=1}^{n}\sum_{i=1}^{n_e} N_i(s)\cdot {\vartheta_t}_i^{\,l}\cdot\cos\left(\frac{l\pi}{\alpha}\theta\right). \end{aligned} \tag{5}$$

Here n denotes the number of harmonic terms, n_e the number of nodal lines per element and α is the curved coordinate. The important thing to note here is that in a FSM model, the nodal amplitudes attached to a particular harmonic term take the role of nodal unknowns, and so the state variables. From the expressions of (3), (4) and (5), one can deduce the expression of the potential energy of the structure. Finally, the discretized equilibrium equation can be obtained by minimizing the total potential energy, with respect to all nodal amplitudes. After carrying out this, we will come to a set of linear equations, which can be best represented in matrix form. Therefore:

$$\begin{bmatrix} [K^{11}] & & & [0] \\ & [K^{22}] & & \\ & & \ddots & \\ [0] & & & [K^{nn}] \end{bmatrix} \begin{Bmatrix} \{a^1\} \\ \{a^2\} \\ \vdots \\ \{a^n\} \end{Bmatrix} = \begin{Bmatrix} \{f^1\} \\ \{f^2\} \\ \vdots \\ \{f^n\} \end{Bmatrix}. \tag{6}$$

Thanks to the special case of boundary conditions and the orthogonal property of the trigonometric functions, the system matrix has a block-diagonal structure, which allows us to solve the equations for each harmonic separately. After all it can be diagnosed, that the FSM can be economically competitive with the FEM in the case of special boundary conditions, or if the span length is considerably large.

3 THE OPTIMALITY CRITERIA METHOD (OC)

As previously mentioned, the sufficient criteria for a design to be a local optimizer are called KKT conditions and can be regarded as a generalisation of the Lagrange multiplier technique. A cardinal point is the selection of the design variables. For plated structures the thickness can play the role, but more often an artificial material model is adopted in the following form:

$$E(\rho) = \rho \cdot E_0.$$

Here ρ is an artificial density, taking values between 0 and 1, E_0 is the Young's modulus. Thus we can preamble a parameter, whereby the stiffness matrix shows linear dependence. This type of formulation is called a SIMP method. In the special case, when the stiffness matrix shows linear dependence from another parameter, that can be chosen as the design variable as well, but this is valid only for certain cases. In the case of nonlinear dependency, the convergence of the shortly introduced method cannot be generally guaranteed.

Our goal is to minimize the cost function, so the volume of the structure. We define two constraints on the state variables. The first one is to satisfy the discretized equilibrium equations, the second one is the already introduced upper bound on the compliance of the structure. This bound is practically the multiple value of the compliance of the initial design, and is denoted later by C_0. The optimization problem in the case of 2D problems and using the FEM as the analysis tool:

$$\begin{cases} \min\limits_{\rho} V(\rho) = \sum_{i=1}^{N} A_i \cdot t_i \cdot {\rho_i}^{\frac{1}{p}}, \\ subject\ to \quad \{\mathrm{u}\}^{\mathrm{T}}[\mathrm{K}]\{\mathrm{u}\} \leq C_0, \\ \quad and \qquad [\mathrm{K}]\{\mathrm{u}\} = \{\mathrm{f}\}. \end{cases} \tag{7}$$

The p parameter in the cost function fastens the interation, its value is recommended between 1 and 3. It has an effect of obtaining a 0/1 type density distribution at the end of the iteration process, which is fundamental, since intermediate values make no real sense. The last equation in (7) is the equilibrium equation, which will be handled separately, as an inner step of an iteration cycle. By writing the Lagrange function, we can transform the problem to the minimization of an unconstrained one.

$$L(\{\rho\}, \lambda) = V(\rho) - \lambda \cdot (\{\mathrm{u}\}^{\mathrm{T}}[\mathrm{K}]\{\mathrm{u}\} - C_0) \tag{8}$$

Here the original restriction on the compliance is written as equality. Without the details we mention, that the equivalence of the two formulations could be admitted by the use of slack variables. From now on we are searching for the minimizer of (8), which is the function of the design variables $\{\rho\}$ and the Lagrange multiplicator λ. Summarizing, the KKT conditions state that for a design to be a minimizer has to satisfy the followings:

$$\begin{gathered} \nabla_{\{\rho\}} L(\{\rho\}, \lambda) = 0, \\ \lambda \geq 0, \\ {\{\mathrm{u}\}_i}^{\mathrm{T}}[\mathrm{K}]_i\{\mathrm{u}\}_i \leq C_0, \\ \lambda\left({\{\mathrm{u}\}_i}^{\mathrm{T}}[\mathrm{K}]_i\{\mathrm{u}\}_i\right) = 0, \end{gathered} \tag{9}$$

with the comment, that for these conditions to be sufficient, it is necessary that the Hessian of the cost function is positive definite. Another remark is that the case $\lambda = 0$ means, that the minimizer coincides with that of the original cost function of (2) without any constraints, so it can be excluded. From the rest, one can obtain closed formulas for the design variables and λ:

$$\rho_i = \left(\frac{p \cdot \lambda \cdot R_i}{A_i \cdot t_i}\right)^{\frac{p}{1+p}}, \quad \lambda = \left(\frac{C_A}{C_0 - C_P}\right)^{\frac{1+p}{p}}, \tag{10), (11}$$

where

$$R_i = \rho_i {\{\mathrm{u}\}_i}^{\mathrm{T}}[\mathrm{K}]_i\{\mathrm{u}\}_i \text{ and} \tag{12}$$

$$C_A = \sum_{active}\left[\left(\frac{A_i \cdot t_i}{p}\right)^{\frac{p}{1+p}} \cdot R_i^{\frac{1}{1+p}}\right], \quad C_P = \sum_{passive}\frac{R_i}{\rho_i}. \tag{13}$$

A strip is active, if the attached design variable has an intermediate value, otherwise it is passive. The details of the deduction of (12) and (13) can be found in [10], while the steps of one iteration cycle is illustrated on the flowchart of Figure 5.

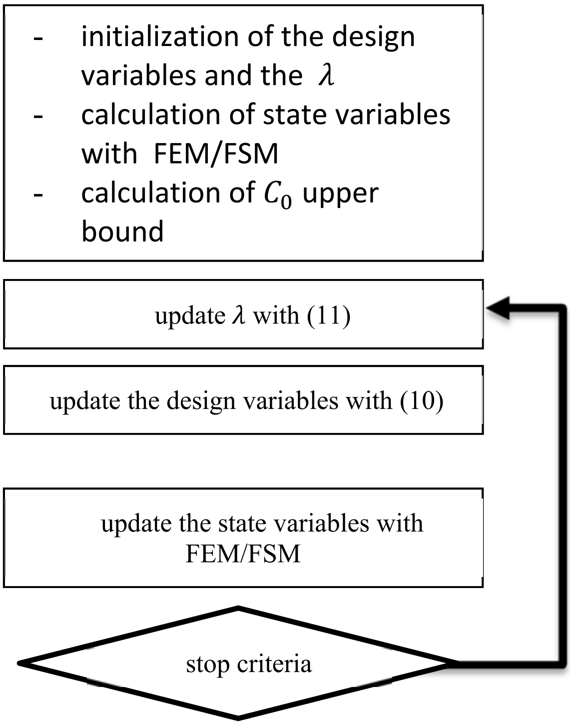

Fig. 5: SIMP flowchart

If we use the FSM for the calculation of the state variables, the problem of (7) modifies to

$$\begin{cases} \min\limits_{\rho} V(\rho) = \sum_{i=1}^{N} A_i \cdot t_i \cdot \rho_i^{\frac{1}{p}}, \\ subject\ to \quad \{\mathrm{a}\}^{\mathrm{T}}[\mathrm{K}]\{\mathrm{a}\} \leq C_0, \\ \quad and \quad [\mathrm{K}]\{\mathrm{a}\} = \{\mathrm{f}\}, \end{cases} \tag{14}$$

as the nodal amplitudes take the role of state variables.

4 OPTIMIZING WITH THE FINITE ELEMENT METHOD

The available literature is sufficiently rich in result obtained with a SIMP method using the FEM, so just a few remarks is to be added here.

The following examples are shown on a rectangular sheet with side length of 8 and 6 meters in x and y directions respectively. A fix support is placed on the left edge, the load is a concentrated one of 100 kN pointing downwards, positioned in the top right corner. The material parameters are equal to that of a C12/15 concrete (E=2600 MPa, ν=0.2). The optimal topology of this setup is shown on Figure 6 with different elements.

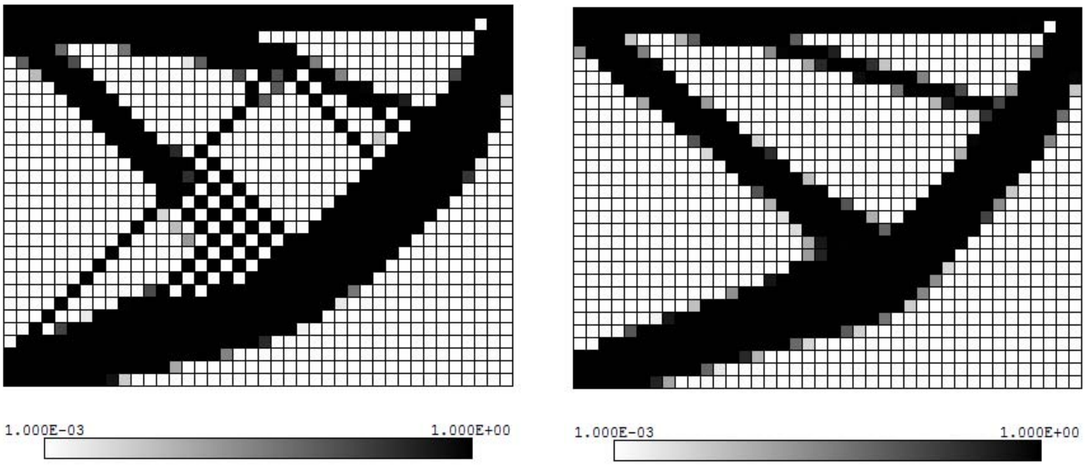

Fig. 6: Optimal results using a four nodded (left) and a special finite element with drilling degree of freedom (right)

At the picture on the left, one can see the so called ‘checkerboard pattern’, which is caused by the incorrect approximation of the FEM. Solutions to this problem are known. One of them is the application of shape functions of a higher order, but it comes with a higher number of variables. The most popular way to avoid this error is nowadays the application of filters of different types. The common property of these is that they make subsequent amendments on the pattern at every iteration step. In this article we suggest another way of thinking by adding a rotational nodal freedom around the *z* axis, making the model able to take into account the infinitesimal contact surface between elements in the case of a checkerboard pattern. The implementation of an extra rotational degree of freedom around the third axis is not rare at plate or shell problems, but their only purpose is to facilitate the transformation from local to global, when adjacent nodes are not coplanar, without taking any real part in the formulation of the stiffness.

From the models containing a drilling degree of freedom we adopted the version educed by Ibrahimbegovic [5], which is based on the variational formulation of Hughes and Brezzi [6], combined with an Allman-type interpolation field for in-plane displacements and the standard bilinear independent normal rotation field [5]. The latter one has the following form:

$$\vartheta_z = \sum_{i=1}^{4} N_i(\xi,\eta)\vartheta_{z_i}, \tag{15}$$

and the Allman-model for the interpolation of the in plane displacement fields:

$$\{\mathrm{u}\} = \begin{Bmatrix} \mathrm{u} \\ v \end{Bmatrix} = \sum_{i=1}^{4} N_i(\xi,\eta) \begin{Bmatrix} \mathrm{u}_i \\ v_i \end{Bmatrix} + \frac{1}{8}\sum_{k=5}^{8} N_k(\xi,\eta)(\vartheta_{z_j} - \vartheta_{z_i}) \begin{Bmatrix} \mathrm{y}_{ij} \\ x_{ij} \end{Bmatrix}, \tag{16}$$

where

$$x_{ij} = x_j - x_i,\ y_{ij} = y_j - y_i, \tag{17}$$

$$N_i(\xi,\eta) = \frac{1}{4}(1+\xi_i\xi)(1+\eta_i\eta) \quad i = 1,2,3,4, \tag{18}$$

$$N_k(\xi,\eta) = \frac{1}{2}(1-\xi^2)(1+\eta_k\eta) \quad k = 5,7, \tag{19}$$

$$N_k(\xi,\eta) = \frac{1}{2}(1-\eta^2)(1+\xi_k\xi) \quad k = 6,8, \tag{20}$$

and the ordered triplets (k, i, j) are defined by (5, 1, 2), (6, 2, 3), (7, 3, 4) and (8, 4, 1) in the case of the standard numbering scheme of element nodes. According to the variational formulation of Hughes Brezzi, the element stiffness matrix can be composed as:

$$[K_e] = [K_{e_m}] + [P_\gamma] = \int_\Omega [B_e]^T [D_m][B_e] d\Omega + \gamma \int_\Omega \{b\}^T \{\mathrm{b}\}\, d\Omega. \tag{21}$$

Here $[D_m]$ is the material stiffness tensor of the sheet, $[B_e]$ is the geometrical matrix of the Allman-type model, γ penalty parameter, and {b} is related to the skew-symmetric part of the strain tensor, in details:

$$[B_e] = \begin{bmatrix} N_{i,x} & 0 & Nx_{i,x} \\ 0 & N_{i,y} & Ny_{i,y} \\ N_{i,y} & N_{i,x} & Nx_{i,y} + Ny_{i,x} \end{bmatrix}, \tag{22}$$

$$\{\mathrm{b}\}^T = \begin{Bmatrix} -\frac{1}{2} N_{i,y} \\ \frac{1}{2} N_{i,x} \\ \frac{1}{16}\left(-\mathrm{y}_{ij} N_{l,y} + \mathrm{y}_{ik} N_{m,y} + \mathrm{x}_{ij} N_{l,x} - \mathrm{x}_{ik} N_{m,x}\right) - N_i \end{Bmatrix}, \tag{23}$$

where

$$Nx_i = \frac{1}{8}\left(\mathrm{y}_{ij} N_l - \mathrm{y}_{ik} N_m\right), \tag{24}$$

$$Ny_i = \frac{1}{8}\left(\mathrm{x}_{ij} N_l - \mathrm{x}_{ik} N_m\right). \tag{25}$$

In expressions (24) and (25) the indices i, j and k are ordered triplets, which can be deduced from the interpolation model and the node numbering of the element. The value of the penalty parameter is problem dependent, usually to match the bounds $\gamma/G = 1$ and $\gamma/G = 10000$. According to [7], [8] and [9] a value of G (shear modulus) is taken.

5 OPTIMIZING WITH THE FINITE STRIP METHOD

Comparing (7) and (14) it can be seen, that if using the FEM or the FSM, there is no difference in the mathematical formulation, but in the interpretation of the results, as explained in this chapter. To obtain a topology optimization algorithm with the FSM we have to ensure that the target domain is well discretized and that the design variables can take the zero value.

The results are illustrated on a girder bridge with a box section, having a cross section with overall dimensions of 3x1 meters and a span length of 8 meters. The bottom flange is 1.5 meters wide and the thickness is 10 cm in general. The material parameters are equal to that of a C12/15 concrete (E=2600 MPa, ν=0.2). Two concentrated forces are applied with a value of -100 kN in vertical direction, positioned at the midspan, as show on Figure 7. The boundary conditions are simply supported hinges at the strip ends.

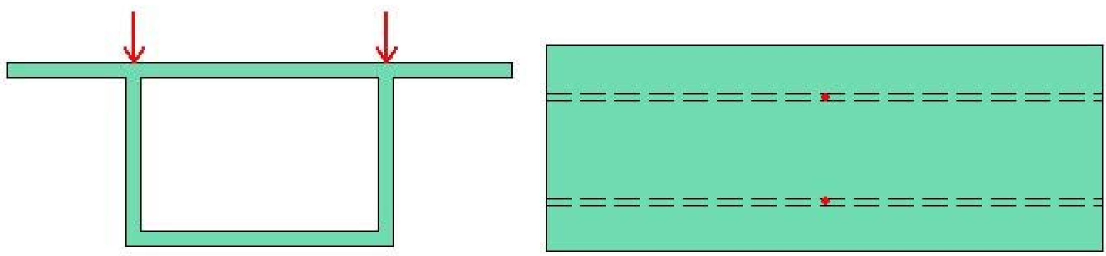

Fig. 7: Setup of a box girder bridge

The design space, which is illustrated on the left of Figure 8, was defined from practical viewpoints. The strips with thick line are kept on a constant density value of 1, while some strips are excluded from the feasible domain. The optimized design can be seen on the right of the same figure. It gives a valuable insight to the optimal shape of the structure, but is not suitable for design.

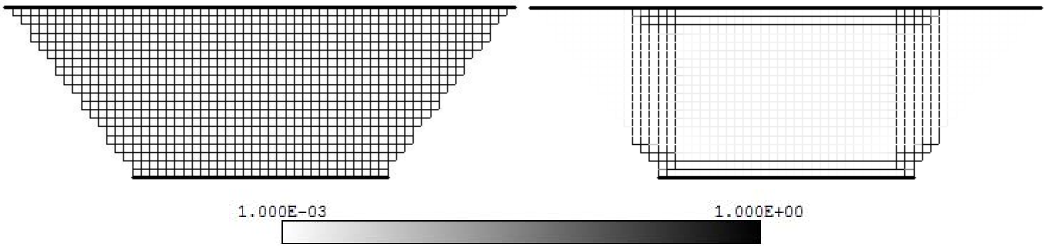

Fig. 8: Optimized section of a box girder bridge

6 CONCLUSION

The results obtained with a FEM based SIMP method is well suited for design, and the results are well known. Within this topic we suggested an alternative method to avoid a specific numerical error, which has proven to be effective yet.

Despite the general applicability of FEM, and in the case of some regularity in the geometry, the system variables can be reduced significantly when using the FSM instead. This semi-analytic method had just partly lived up to expectations. The restriction of the classical finite strip formulations to the boundary conditions is already resolved, but the results are hard to interpret and utilize in the design of structures.

ACKNOWLEDGMENT

This project has been completed thanks to the financial support provided by the Hungarian Research Fund. Registration number of the project is 81185.

REFERENCES

[1] VENKAYYA, V. B., Khot, N. S., BERKE, L. Application of optimality criteria approaches on automated design of large practical structures. In *Proc. 2nd Symp. on Structural Optimization, AGARD-CP-123*, Milan, Italy, pp. 3.1–3.19, 1973.

[2] BERKE, L., VENKAYYA, V. B. Review of optimality criteria approaches to structural optimization. In J. L. A. Schmit (Ed.) *Proc. Structural Optimization Symp., ASME AMD*, Vol. 1. pp. 23-24. 1974.

[3] DOBBS, M. W.; R. B. NELSON. Application of optimality criteria to automated structural design. *AIAA J.*, 1975, vol. 14, issue 10, pp. 1436-1443.

[4] HEGEMIER, G.A.; PRAGER, W. (1969): On Michell Truss, Int. J. Mech. Sci., 11, 209-215

[5] IBRAHIMBEGOVIC, A.; TAYLOR, R. L.; WILSON, E. L. A robust quadrilateral membrane finite element with drilling degrees of freedom. *International Journal for Numerical Methods in Engineering*. 1990, vol. 30, issue 3, pp. 445–457.

[6] HUGHES, T. J. R.; BREZZI, F. On drilling degrees of freedom. *Computer Methods in Applied Mechanics and Engineering*. 1989, vol. 72, issue 1, pp. 105–121.

[7] ALLMAN, D. J. A compatible triangular element including vertex rotations for plane elasticity analysis. *Computers and Structures.* 1984, vol. 19, issue 1, pp. 1-8.

[8] LONG, C. S.; GEYER, S.; GROENWOLD, A. A. A numerical study of the effect of penalty parameters for membrane elements with independent rotation fields and penalized equilibrium. *Finite Element in Analysis and Design.* 2006, vol. 42, issue 8-9, pp. 757–765.

[9] LIU, J.; RIGGS, H. R.; TESSLER, A. A four-node, shear-deformable shell element developed via explicit Kirchhoff constraints. *International Journal for Numerical Methods in Engineering*. 2000, vol. 49, issue 8, pp. 1065–1086.

[10] LÓGÓ, J. New Type of Optimal Topologies by Iterative Method, *Mechanics Based Design of Structures and Machines*. 2005, vol. 33, issue 2, pp. 149-172.

Reviewers:

Prof. Ing. Zdeněk Kala, Ph.D., Institute of Structural Mechanics, Faculty of Civil Engineering, Brno University of Technology, Czech Republic.

Doc. Ing. Katarína Tvrdá, PhD., Department of Structural Mechanics, Faculty of Civil Engineering, Slovak University of Technology in Bratislava, Slovakia.