Journal of the Serbian Society for Computational Mechanics / Vol. 1 / No. 1, 2007 / pp. 99-109

Application of Link FE in Modeling of Specific Boundary/Interface Conditions

D. Kovačević[1], R. Folić[2], Đ. Lađinović[3]

[1, 2, 3] Civil Engineering Department, Faculty of Technical Sciences,
University of Novi Sad, Trg D. Obradovića 6, 21000 Novi Sad, Serbia
[1]e-mail: dusan@uns.ns.ac.yu; [2]e-mail: folic@uns.ns.ac.yu; [3]e-mail: ladjin@uns.ns.ac.yu

Abstract

This paper presents the concept of so-called "link" finite elements (link FE) application in finite element method (FEM) modeling of civil engineering structures. Some theoretical and numerical aspects as well as modeling performance of link FE are discussed.

The basic approach considers the FEM structural modeling in simplified sense: after FE mesh generation, i.e. system topology definition, follows modeling the FE stiffness and theoretical boundary/interface conditions. Advanced approach, proposed in this paper, takes into account real boundary and interface conditions and some particular phenomena modeling.

Required theoretical considerations, suggestions and recommendations for practical link FE application in FEM modeling are given. Also, importance of Computer Aided Structural Analysis (CASA) software choice is emphasized.

As an illustration of the theoretical considerations, four numerical examples are given. The first example shows possibility of modeling the composite beam. Second example presents effects in plate behavior when link FE are used. Third example explains behavior of "sandwich-plate" formed by a three layers. Fourth example is an illustration of reinforced concrete plate modeling possibility.

Key words: FEM, structural modeling, link FE, boundary/interface conditions

1. Introduction

Due to numerical efficiency and simple software implementation, the FEM has become a dominant method of numerical modeling structural behavior. The FEM modeling is the creation of idealized and simplified representation of structural behavior. Errors and inadequacies in FEM modeling may cause serious design defects and difficulties. The FEM modeling basically comprehend discretization and approximation phase.

User of a FEM software usually has a wide range selection of different FE shapes and types. The FE shape choice is relevant in discretization phase. In the approximation phase of FEM modeling choice of FE type substantially determines quality of the FEM solution.

Discretization (geometrical modeling i.e. generation of a FE mesh) is the initial phase of FEM modeling. Errors of discretization occur due difference between real geometry of a structure and FE system topology (application of inadequate FE shape or insufficient number of FE).

Discretization errors belong to a category which does not essentially change the character of the FEM solution. The proof for this might also be the fact that in the CASA software the generating of FE mesh belongs to the so-called pre-processor. Namely, this operation is possible due to algorithmic description, which is usual for procedures that do not require extremely creative and intuitive approach.

Consequences of unsuitable discretization can be easily observed even by an inexperienced FEM software user and without high level of theoretic knowledge. Contrary, problems that can appear due to errors in approximation (numerical modeling, in strict sense) are much more complex.

2. Modeling of specific boundary and interface conditions

Modeling of boundary and interface conditions is the important step in the FEM approximation phase, because the real behavior of structural system strongly depends of real supports and connections state. In FEM software where an automatic modeling of support and connection behavior is not yet implemented, the structural designer must have knowledge of FEM technology and software implementation.

There are two ways in numerical modeling of the boundary conditions, i.e. setting the specific degrees of freedom (DOF) of the FE system:

- numerical modeling by "skip" and
- numerical modeling by "restraint".

The approach by "skip" is applicable primary for zero value DOF in a support and it is not universal. This approach implies "deletion" of row and column of the global stiffness matrix of FE system for zero DOF. The approach by "restraint" is more comprehensive than the previous one and demands numerical constraint or permission of specific DOF by transformation of the global FE syatem stiffness matrix.

The result is same in both cases: the FE system global stiffness matrix is regular because the so-called "rigid body" DOF are eliminated. If boundary condition is not "standard" (skewed support, for example, Fig. 1), the "restraint" approach demands transformation of the global stiffness matrix, see [2].

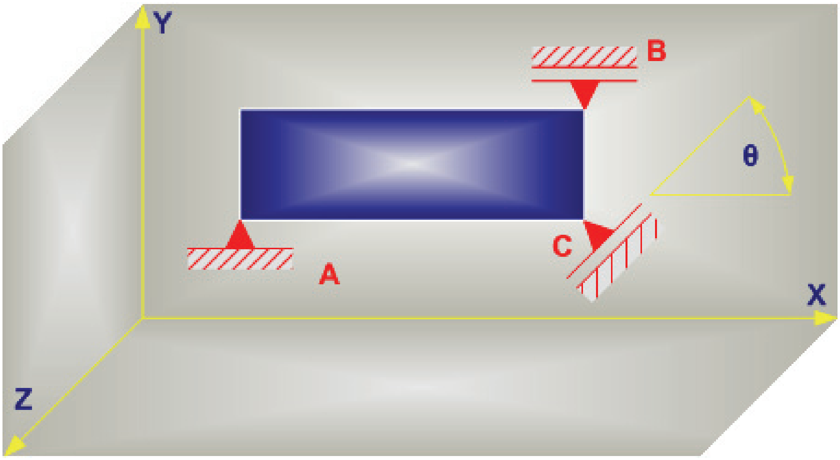

Fig. 1. Standard and nonstandard (inclined by "θ") boundary condition scheme.

Modeling of nonstandard boundary conditions is possible by application of corresponding link FE. They are one-dimensional FE with a selected stiffness characteristic. These FE connect two nodes or two lines and have translation and rotation stiffness components defined in their coordinate system. The "interface" in link FE is determined by geometry of connection (i.e. position of contact) between standard FE (beam, plate, shell). Link FE can have a nonlinear parameter called "limit resistance" that limits the force they are able to transfer.

In some cases inadequate value of link FE stiffness may generate errors in the FEM solution, due to "ill-conditioned" global stiffness matrix (see Fig. 2), for very small stiffness "K_B" with respect to stiffness "K_A".

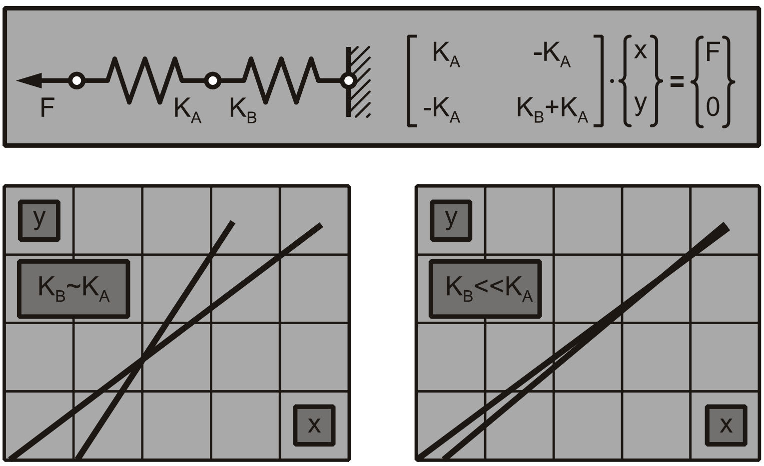

Fig. 2. "Well-conditioned" and "ill-conditioned" FEM solution schemes.

The term "single-joint restraint" is often in use for boundary condition, because the restraint is applied to joint of FE system with respect to support. The so-called "multi-joint restraint" is in use for definition of restraint between two or more FE system joints (interface condition). Apparent example for a "multi-joint restraint" is modeling the slab-beam connection, Fig. 3. In this connection, axis of beam FE and axis of plate FE are not coincident. Eccentricity (i.e. "offset") in local element axes direction are "e_x" and "e_y". In the numerical sense, there is here essential interlock corresponding to DOF of beam FE and plate FE. This interlock is performed by the interface located in the contact ("j" joint). The connection behaviour (rigid, elastic, elasto-plastic) is modeled by the choice of stiffness values and by the rule of stiffness change.

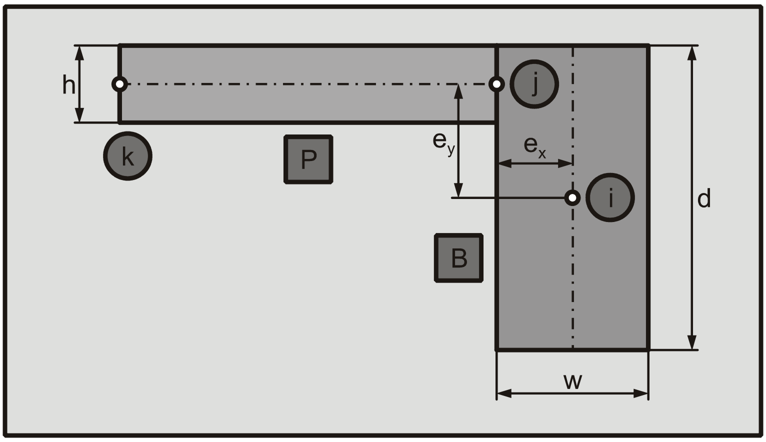

Fig. 3. Example of interface condition i.e "multi-joint restraint".

For totally rigid connection, the matrix relation between corresponding beam DOF ("i" as so-called "master" joint) and plate DOF ("j" as so-called "slave" joint) is:

$$\begin{Bmatrix} u_i \\ v_i \\ \theta_i \end{Bmatrix} = \begin{bmatrix} 1 & 0 & e_y \\ 0 & 1 & e_x \\ 0 & 0 & 1 \end{bmatrix} \begin{Bmatrix} u_j \\ v_j \\ \theta_j \end{Bmatrix} \tag{1}$$

Assembling of a global stiffness matrix of FE system requires two transformations: between "slave" and "master" joint DOF and between DOF in local and global coordinate systems. The "slave" joint DOF do not appear in the global stiffness matrix explicitly. These DOF are computed in a postprocesing phase of the analysis, according to (1).

The interface condition within numerical modeling in the above described approach (the so-called "master-slave elimination") is performed before the assembling of the global stiffness matrix. In the alternative approach in modeling, the interface condition is enforced to the global FEM equilibrium system of equations after the assembling procedure. Two methods are used in the FEM software:

- Lagrange multiplier adjunction, and
- penalty augmentation.

For both methods the restraint equation has the form:

$$\mathbf{Cu} - \mathbf{Q} = \mathbf{0} \tag{2}$$

where: **C** is "*m x n*" matrix with "*m*" as number of restraints and "*n*" as number of DOF,
u is vector of DOF in a global system, and
Q is vector of constants.

Lagrange multiplier and penalty method impose equation (2) on the global equilibrium **Ku** =**R** in a different ways.

Lagrange method introduces additional variables - Lagrange multipliers:

$$\boldsymbol{\lambda}^T = \begin{bmatrix} \lambda_1 & \lambda_2 & \dots & \lambda_m \end{bmatrix} \tag{3}$$

Each restraint equation is in homogeneous form and is multiplied by the corresponding "λ_i":

$$\boldsymbol{\lambda}^T (\mathbf{Cu} - \mathbf{Q}) = 0 \tag{4}$$

The expression for the total energy is obtained when the left side of (4) is added to the typical energy terms:

$$\Pi_p = \frac{1}{2}\mathbf{u}^T\mathbf{Ku} - \mathbf{u}^T\mathbf{R} + \boldsymbol{\lambda}^T (\mathbf{Cu} - \mathbf{Q}) \tag{5}$$

If the derivatives of Π_p with respect to $\mathbf{u}$ and $\boldsymbol{\lambda}$ are set to be equal to zero, it is obtained:

$$\begin{bmatrix} \mathbf{K} & \mathbf{C}^T \\ \mathbf{C} & \mathbf{0} \end{bmatrix} \begin{Bmatrix} \mathbf{u} \\ \boldsymbol{\lambda} \end{Bmatrix} = \begin{Bmatrix} \mathbf{R} \\ \mathbf{Q} \end{Bmatrix} \tag{6}$$

The penalty method is based on equation:

$$\mathbf{t} = \mathbf{Cu} - \mathbf{Q} = \mathbf{0} \tag{7}$$

which implies that the restraints are fulfilled. Similar to (5), the energy expression is:

$$\Pi_p = \frac{1}{2}\mathbf{u}^T\mathbf{Ku} - \mathbf{u}^T\mathbf{R} + \frac{1}{2}\mathbf{t}^T\boldsymbol{\alpha}\mathbf{t} \tag{8}$$

where: $\boldsymbol{\alpha}^\mathbf{T} = [\alpha_1 \; \alpha_2 \dots \alpha_m]$ is the diagonal matrix with "*m*" penalty numbers. If the derivatives of Π_p with respect to $\{\mathbf{u}\}$ are set to zero, it is obtained:

$$\left(\mathbf{K} + \mathbf{C}^T\boldsymbol{\alpha}\mathbf{C}^T\right)\mathbf{u} = \mathbf{R} + \mathbf{C}^T\boldsymbol{\alpha}\mathbf{Q} \tag{9}$$

where: $[\mathbf{C}]^T [\boldsymbol{\alpha}] [\mathbf{C}]$ is the penalty matrix. The penalty numbers must be not too large and not too small, in order to avoid that the FE system matrix becomes ill-conditioned.

3. Numerical examples

As an illustration of previous considerations several numerical examples, using FEM software AxisVM® 8+, see [5], are given. The first example shows possibility of modeling the simple supported composite beam, loaded by uniform unit value vertical load (p=1.00kN/m^2), Fig. 4.

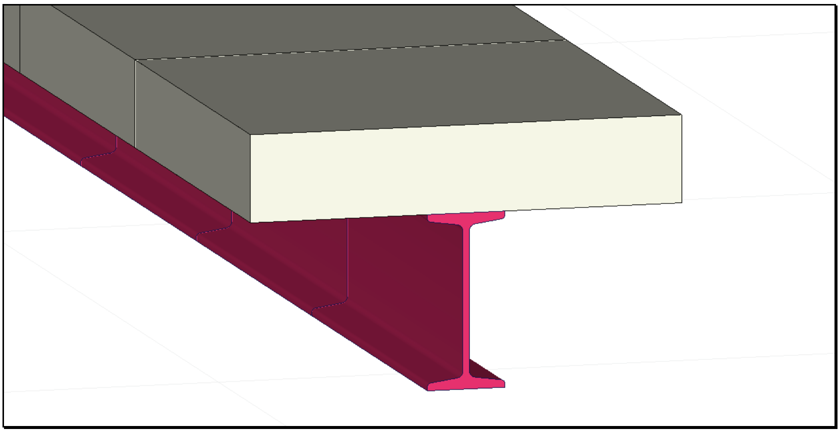

Fig. 4. Numerical example 1: composite beam (steel girder + concrete slab).

Deformation of composite beam modeled by using link FE, without bond (upper panel) and with bond (lower panel), are shown in Fig. 5.

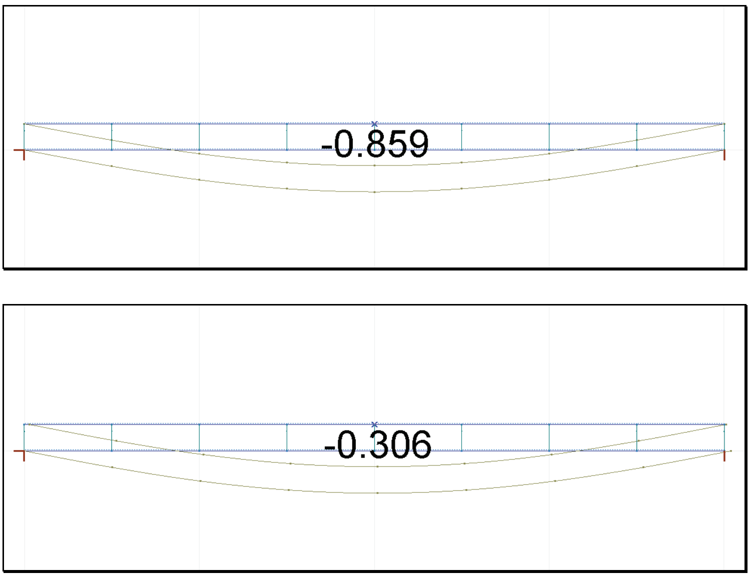

Fig. 5. Beam deflection with the pick values in [*mm*] - modeled by link FE (without bond and with bond).

The second example (Fig. 6) shows plate loaded by uniform unit value vertical load ($p=1.00kN/m^2$). The plate is connected with simple supported beam in two ways, i.e.

- beam axis is in midplane of plate (left panel - case A, without link FE) and
- top surface of plate coincides with top surface of a beam (right panel case B, with link FE).

Figures 7-9 show displacements and moments for both cases.

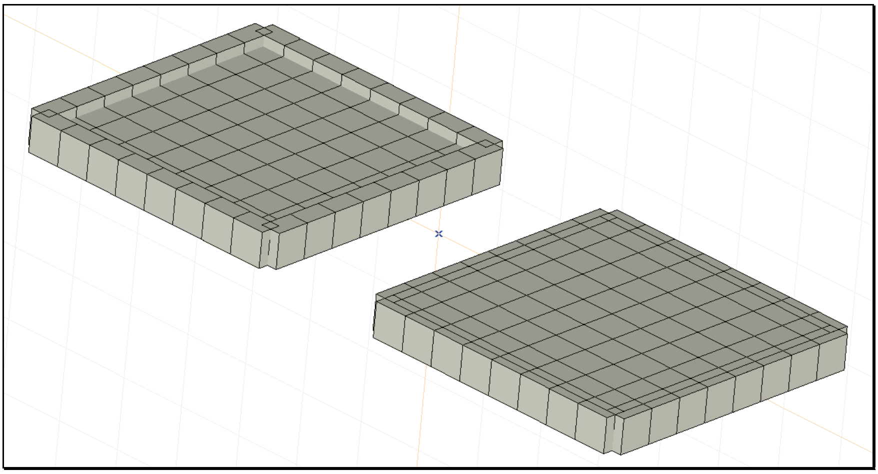

Fig. 6. Numerical example 2: Two cases of plate support by beams:
plate mid-plane and beam axis coincide (left panel, case A);
upper planes of plate and beams coincide (right panel, case B).

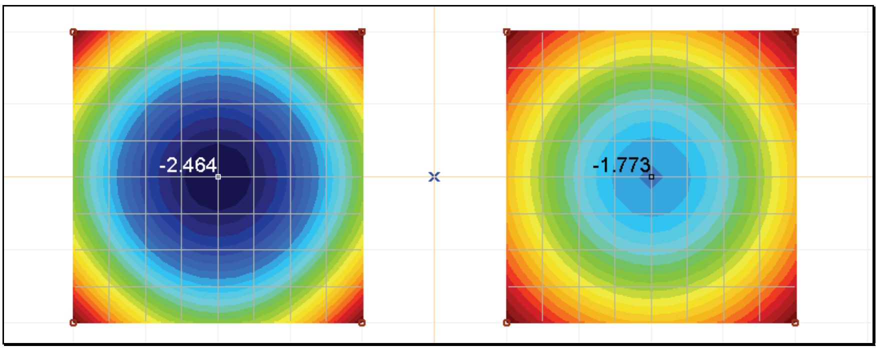

Fig. 7. Distribution of displacements *w* for case A - left panel; and for case B - right panel. The numbers represent the maximum values of displacement.

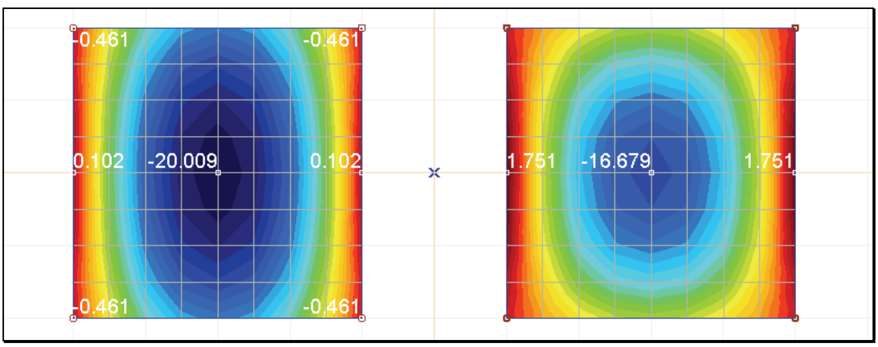

Fig. 8. Distribution of flexural moment M_x for case A - left panel; and case B - right panel.

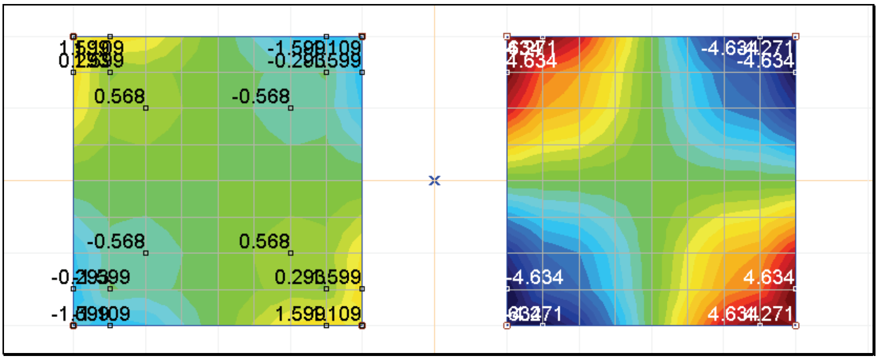

Fig. 9. Distribution of torsional moment M_{xy} for case A- left panel; and case B – right panel.

Difference in maximum displacement (Δw≈40%), flexural moments (ΔM_x≈25%) and torsional moments (ΔM_{xy}≈250%) indicate the necessity of adequate modeling the real connection between plate and beam, especially for beam FE with shear stiffness influence and plate FE which behaves according to Reissner-Mindlin model.

The third example (Fig. 10) shows behavior of the so-called "sandwich-plate" loaded by uniform unit value vertical load ($p=1.00 kN/m^2$) and supported by beams. This composite plate is formed by a three layers: steel foils (top and bottom face) and polystyrene foam core (between steel foils).

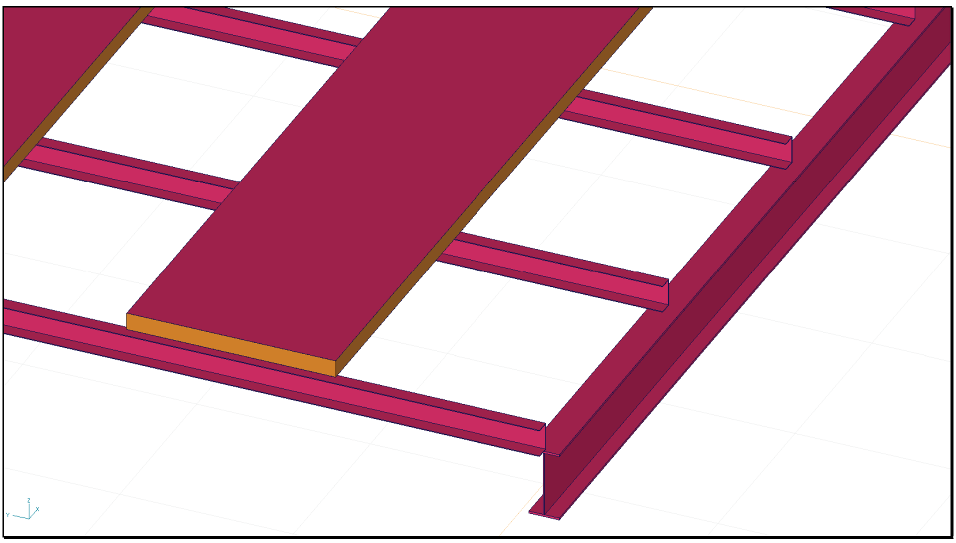

Fig. 10. Numerical example 3: composite “sandwich” plate (steel foil + foam + steel foil).

Stiffness and bearing capacity of this plate is based on shear connection (so-called "bond") between foils and core. Bond deterioration generates stiffness and bearing capacity loss. Modeling of bond is possible by application of link FE. The FE model consists of the top and bottom steel foils, polystyrene foam core and link FE between foils and core, Fig. 11. Modeling of bond behavior is performed by increasing/decreasing stiffness of link FE. Figure 12 shows the peaks of the principal stress in the direction of long plate span in the areas with bond deterioration. This bond behavior is modeled by small value of link FE stiffness (shear component).

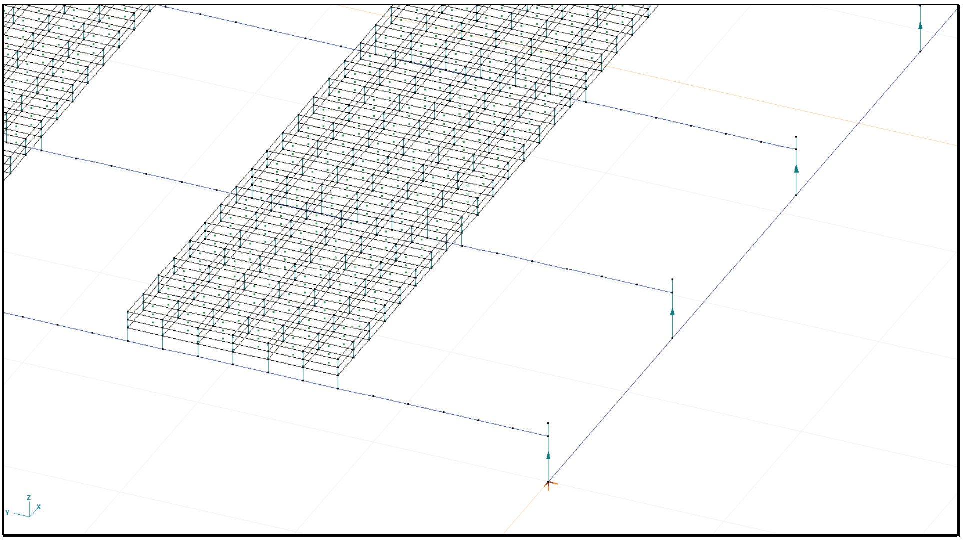

Fig. 11. Modeling of bond between layers of sandwich plate.

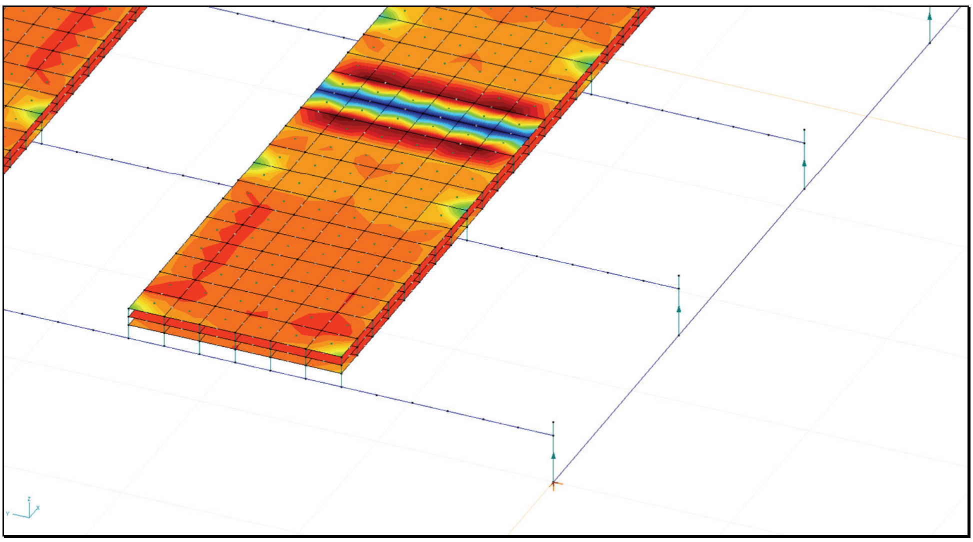

Fig. 12. Principal stress peaks in steel foils as a consequence of bond deterioration.

The fourth example is a reinforced plate, Fig. 13. This plate is simply supported in the short edges and loaded by unit value vertical load ($p=1.00 kN/m^2$). The FE model of plate (Fig. 14) consists of: shell FE for modeling concrete, beam FE for modeling steel bars, link FE for modeling steel-concrete bond and link FE for modeling connection between slab and rib part of plate. Principal stress distribution (in direction of the plate long span) in concrete is given in Fig. 15.

In this example bond between concrete and steel is treated as perfect by choosing large stiffness value at the interface point. Bond deterioration maybe modeled by changing the stiffness of link FE at the interface, according to the corresponding constitutive rule for concrete/steel slip under loading.

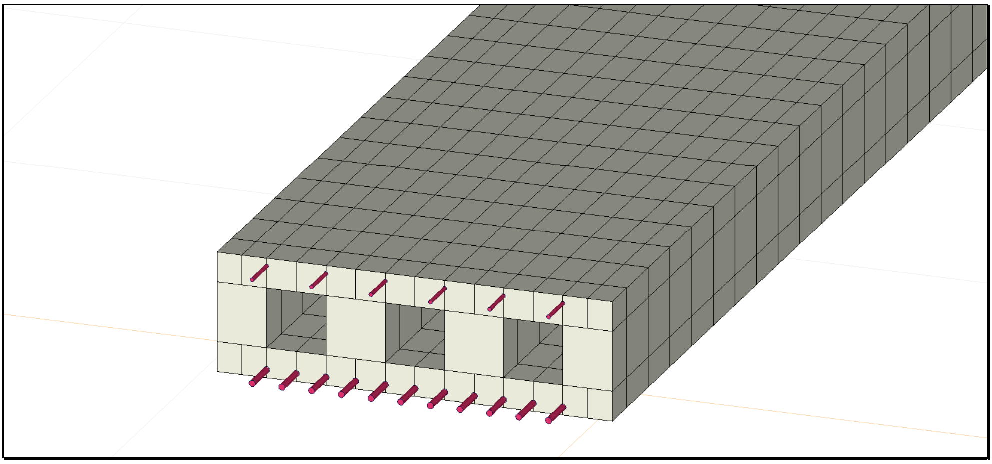

Fig. 13. Numerical example 4: RC floor plate.

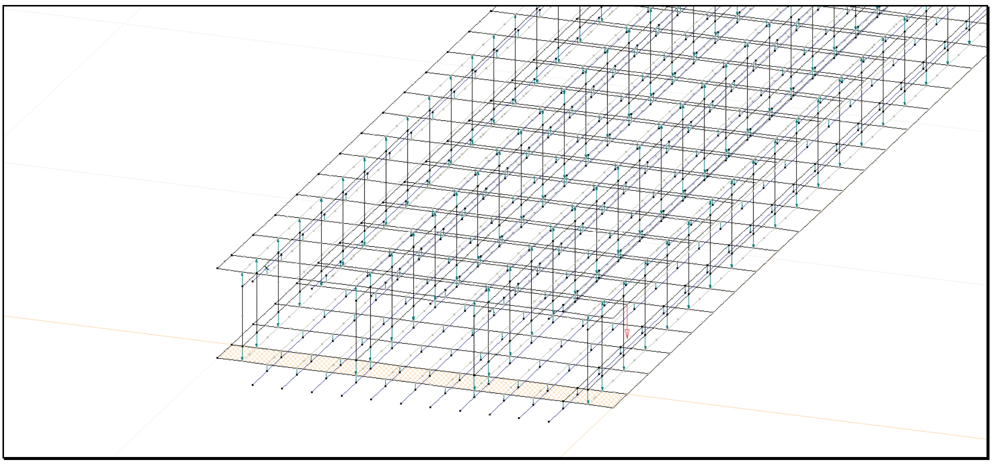

Fig. 14. Modeling bond between concrete and steel in RC floor plate.

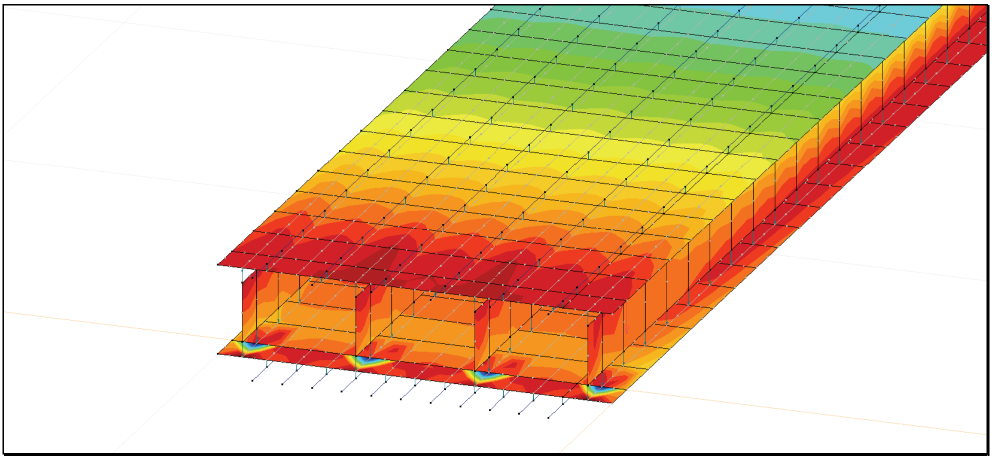

Fig. 15. Principal stress distribution for concrete part of plate in support zone.

Conclusions

This paper emphasized the importance of modeling the boundary and interface conditions in FEM structural analysis. By convenient choice of link FE stiffness parameters it is possible to obtain significantly different behaviors of the modeled structural system (with/without shear stiffness, with/without flexural stiffness, etc.).

The link FEs are important in modeling of reinforced concrete structural elements, eccentrically connected structural elements, cross-section created by different materials, etc.

Adequate implementation of link FEs in a FEM software provides possible improvement in accuracy of the prediction of the structural behavior. Numerical examples are solved using AxisVM® 8+. Besides "node-to-node" link FE, the "line-to-line" link FE for modeling the "wall-plate", "wall-wall" and "plate-plate" connections can be employed using the AxisVM® .

Acknowledgements: This paper is a part of research in Scientific Project Nº6517A granted from Ministry of Science and Environmental protection Republic of Serbia.

References

[1] K.J. Bathe: Finite Element Procedures in Engineering Analysis, Prentice-Hall Inc., New Jersey, 1982.
[2] R.D. Cook: Finite Element Modeling for Stress Analysis, John Wiley & Sons, Inc., 1995.
[3] D. Kovačević: FEM Modeling in Structural Analysis (in Serbian), Građevinska knjiga, Belgrade, 2006.
[4] J.S. Przemieniecki: Theory of Matrix Structural Analysis, McGraw-Hill, New York, 1968.
[5] AxisVM® 8+ Structural Analysis Software - User Manual, InterCAD, Budapest, 2006.