Hybrid Models for Efficiently Analysing Overall and Local Stability Problems in the Bridge Design Process

Dorian Janjic
Civil Engineer
TDV GmbH/Bentley
Graz, Austria
Dorian.Janjic@Bentley.com

Dorian Janjic, born 1960, civil engineering degree from the Faculty of Civil Engineering, Sarajevo. 20 years of experience in technical research, software development and project work.

Johann Stampler
Civil Engineer
TDV GmbH/Bentley
Graz, Austria
Johann.Stampler@Bentley.com

Johann Stampler, born 1951, civil engineering degree from Graz University of Technology. More than 30 years of experience in structural analysis and Finite Element techniques.

Heinz Bokan
Manager Project Centre
TDV GmbH/Bentley
Graz, Austria
Heinz.Bokan@Bentley.com

Heinz Bokan, born 1947, civil engineering degree from the Graz University of Technology. More than 30 years of experience in structural analysis in a wide range of applications.

Summary

This contribution presents a novel approach for bridge design analyses. The full 3D design analysis is based on a hybrid model combining advantages of the classical beam theory and the Finite Element Method. With defining few additional parameters, the deformability of the cross-section can be taken into account by using Finite Element techniques up to any required level of accuracy. Instead of presenting FE results - like stresses in Gauss points – this novel approach provides integral "beam-like" results for each segment face. The user can freely choose the number of integral result. This allows for applying standard design check routines in the post-processing phase in accordance with current design codes, which are usually based on "beam" results.

Keywords: Bridge analysis, Beam elements, Finite Elements, Hybrid approach, Stress distribution, Integral results, Proof checks

1. Introduction

Bridge engineers are used to classical beam theory in their design work. Therefore most structural analysis software packages for bridge engineering have been based on beam elements. I.e. they are basically space-frame programs, where the beam theory applies for the individual elements. This theory assumes that cross-sections remain plane and un-deformed.

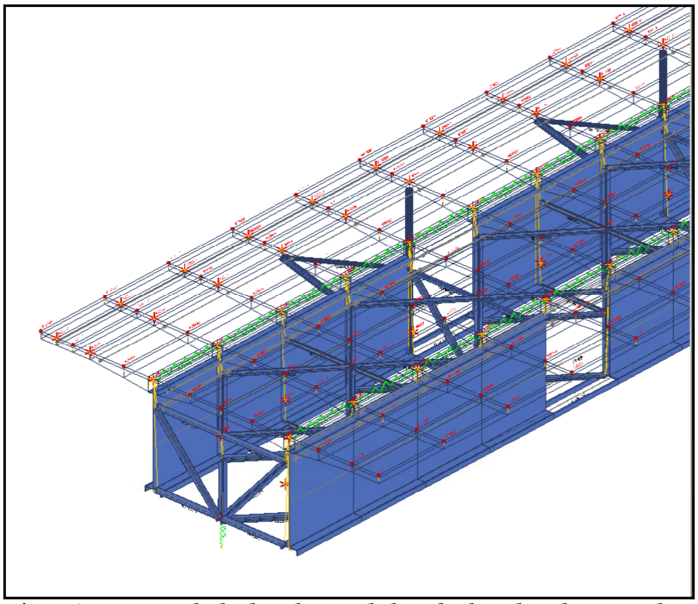

Fig. 1 *Typical hybrid model of the bridge with beam steel members and "virtual" beam slab implemented as substructure with shell elements*

Additionally, we have the assumption of plane stress conditions.

In practice, the approach of considering beam elements has the advantage that all design code regulations relate to respective beam element results. That means, that the results of the program directly allow for doing the required proof checking processes in accordance with the relevant design codes.

However, the structural behaviour of bridge superstructures shows often effects, which cannot be properly taken into account with using the classical beam theory.

Therefore, the strong wish has widely arisen to use a continuum mechanics approach, i.e. particularly Finite Element techniques, for solving these problems.

There are mainly 2 different approaches to improve the accuracy of the analysis. The first uses a modelling with more beam elements for one section – equivalent grid models – or extensions to the beam theory introducing warping degrees of freedom [1]. The second approach is based on direct usage of plate and shell elements [2] in bridge analysis.

The presented new approach is a combination of the traditional beam analysis with the possibility, to consider cross-section deformations by using Finite Element techniques. This implementation has the advantage that the user creates his calculation beam model as he is used to do. With few additional parameters the deformability of the cross-section is taken into account (Figure 1). The user may refine his model up to any level of accuracy without loosing integral section forces.

Thus, using this FE approach saves the tedious work for defining a fictitious girder grid model for approximately taking into account the deformability of the cross-section. Both advantages of the beam model and of the shell and plate elements are fully used.

2. Basics of the FEM implementation

2.1 The Challenge

The FE method has been successfully used for many years for solving continuum mechanics problems in engineering. The primary result besides the deformation state is the stress field in the structure. However, in bridge engineering and other fields of structural engineering, the stresses are not sufficient for suitably assessing the bearing behaviour and stability of the structure. An integral approach dealing with internal forces rather than stresses is generally required. Stress peaks can often be accepted as local phenomena, whereas internal sectional forces being the integral of the stresses over the section determine the safety of the structure.

Complex post-processing functions are therefore required in traditional FE analyses in order to allow for doing proof checks in accordance with the design codes. This includes sophisticated interpolation, extrapolation, averaging and integration procedures in order to get the required result basis in terms of internal forces and moments. The real challenge is to get a more accurate model than in traditional beam analysis without major additional model preparation effort and without loosing the required result basis. The new approach therefore addresses the following:

- Simple data preparation is essential; otherwise engineers will not use the refined approach in practice
- The approach must allow for mixing traditional beam elements with any type of finite elements, i.e. full compatibility between different element types must be guaranteed.
- The approach must support the full functionality of the traditional beam solution, i.e. special functions derived for beam elements must be implemented for shell or volume elements.
- The result presentation must provide local FE results like stress distributions in cross-sections as well as integral results.

2.2 Hybrid FE model – Step by Step Model Refinement

The requirement for simple data preparation immediately yields the demand, that basic preparation work shall be done as common practise in terms of beam models, i.e. the superstructure is modelled

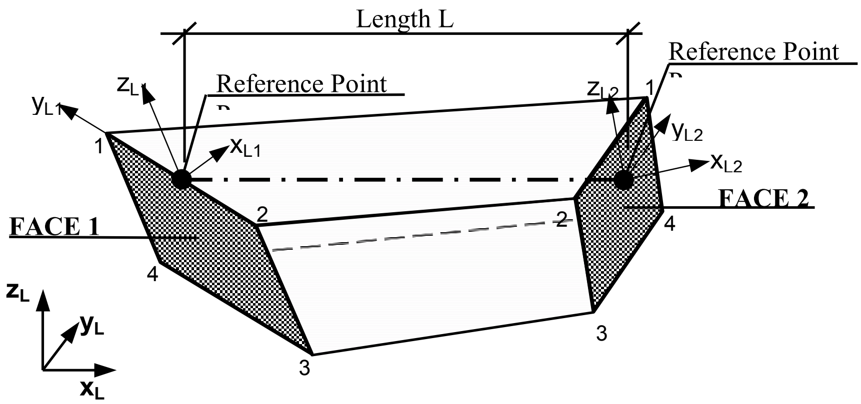

Fig. 2 *Structural segments defined by the length, and the face geometry at begin and end*

with beam elements arranged along the bridge axis, and the stiffness of these elements is described by the cross-section properties.

In the hybrid model the basic structural object is a non-prismatic *segment* with two or more faces. As shown in Figure 2, this segment can be modelled in a simplified way – as beam element – or more accurately as a set of Finite Elements acting as substructure. In the second case all "beam like" results and properties still exist and the segment acts as "virtual beam" although the analysis itself uses the stiffness properties of the substructure.

An automatic procedure is implemented to directly deriving the FE substructure model from the equivalent beam model. This allows for instance for using a standard beam model in the preliminary design state and for improving the accuracy of the model step by step in accordance with the requirements of the design process as shown in Figure 3.

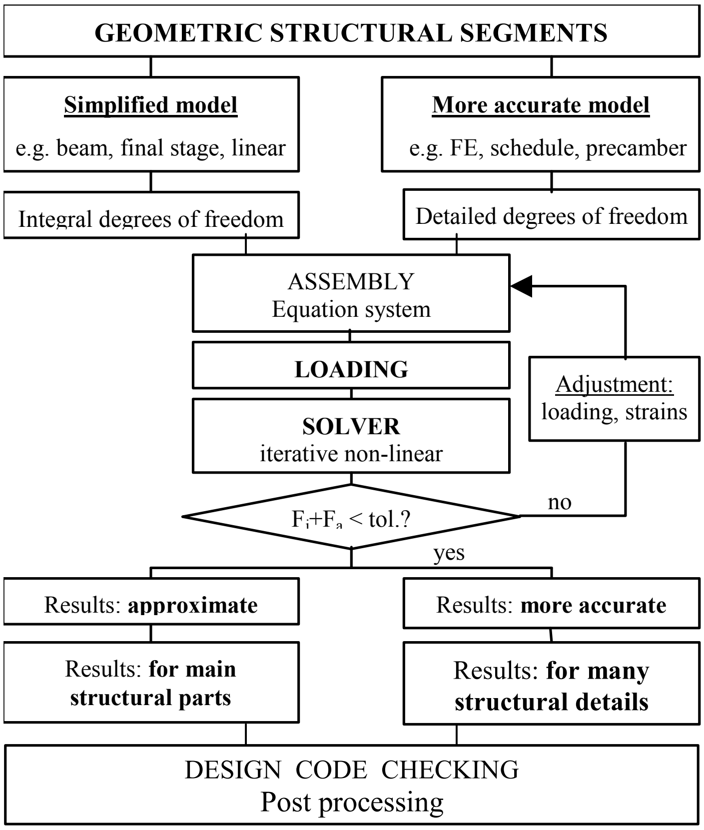

Fig. 3 *Flow chart of hybrid FE analysis*

With this comprehensive hierarchical approach where the FEM model is directly derived from an equivalent beam model, it is an easy step forward to offer the possibility for using integral results in the post-processing phase in addition to considering the stress field and the local effects. The stresses are integrated over the cross-sections and can be used in the design code checks in the standard manner.

2.3 Compatibility Problems

It is well known that the nodes of classical shell elements don't have corresponding stiffness for "in plane" rotation, often called "drilling degree of freedom". This produces difficulties for using shell elements together with beam elements. Even the simplest case of linking a standard 3D beam with a shell element fails.

Different solutions for this problem are possible – from introducing a dummy stiffness to creating special transition elements. However, the best solution for practical usage is to naturally incorporate this stiffness the shell elements.

The presented approach uses this idea. Based on a variational formulation, a very robust shell element is implemented as proposed in references [3] to [9]. This element has a drilling stiffness incorporated, which is compatible to the rotational stiffness of any connected beam element.

The example of Figure 4 illustrates this. In this example, the loading has been applied as concentrated beam moment in the middle of the span, which is fully clamped at both ends. Both, the simple beam model (right) and the FE model (left) show the behaviour very clearly.

2.4 Rigid Links as natural DOF constraints

Structural nodes correspond to the point in 3D place where the equilibrium of discretized system is checked. As various FE types are assembled together, distances between element nodes and global nodes may exist. Therefore "rigid links" are necessary to properly assemble the structural model.

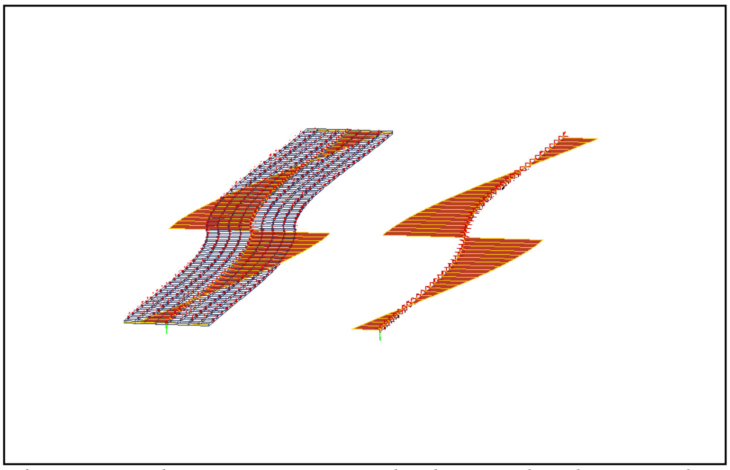

Fig. 4 *In plane moment applied as a loading to the plate bridge modelled with shell elements*

As the reduction of DOF in the structural model becomes important due its size it is preferred to fully constrain nodes from the system of equations. In order to establish full 3D model up to the desired complexity it is necessary to allow for rigid links for all FE types. Although this extension increases computational time for element stiffness, damping and mass matrix, the benefits from this modelling feature are so strong that in this work all elements are extended with rigid links constraints.

However, implementation of automatic rigid link constraint has to be properly implemented. Special care is needed for dynamics effects – where both velocity and acceleration nodal values have to be constrained in the same matter - and for large displacements problems including even simple p-delta effects. Implementation of rigid links without real rotation model produces inaccurate results and therefore real rotations model is used in this work, resulting in no strain field if rigid body movement is applied to the structural element.

For solution of large displacement problem, the co-rotational formulation is chosen in this work for both linear and non-linear finite elements as the optimal one for large-rotation small-strain analysis. This allows for common usage of engineering strain – small strain analysis – in all problems.

3. Data Preparation

3.1 Definition of Finite Elements

As already stated in 2.1, a simple data preparation is an absolute must, otherwise engineers will refuse using the refined approach and accept the lack of accuracy, often combined with a lack of safety due to not considering possibly dangerous effects of the bearing behaviour and failure modes.

In the new approach, finite element definitions are primarily done in the geometric pre-processor of the program. The user applies the standard input process, i.e. the cross-sections are defined, and an axis, where the cross-sections are aligned along. The cross-sections consist of finite elements, which have already also been used in beam models for calculating the cross-section values of the beam elements.

The first step in the new approach is the assignment of the element types for these elements being used as finite elements in the analysis process. These elements can be wall elements, plate elements, shell elements or solid elements. The only extension is, that the user be aware that using the cross-section elements as finite elements in the analysis process possibly requires a finer meshing than required for calculating the cross-section values.

Figure 4 shows exemplarily the simplicity of the input process on behalf of a girder with corrugated steel web, where the influence of the accordion effect is investigated. The whole structure is described by two cross-sections, the rectangle concrete end cross-section, and the composite cross-section with concrete flanges, steel flanges and the steel web with variable position of the centre face with respect to the centre line of the flanges. This variable value is simply described as a function of the station in longitudinal direction.

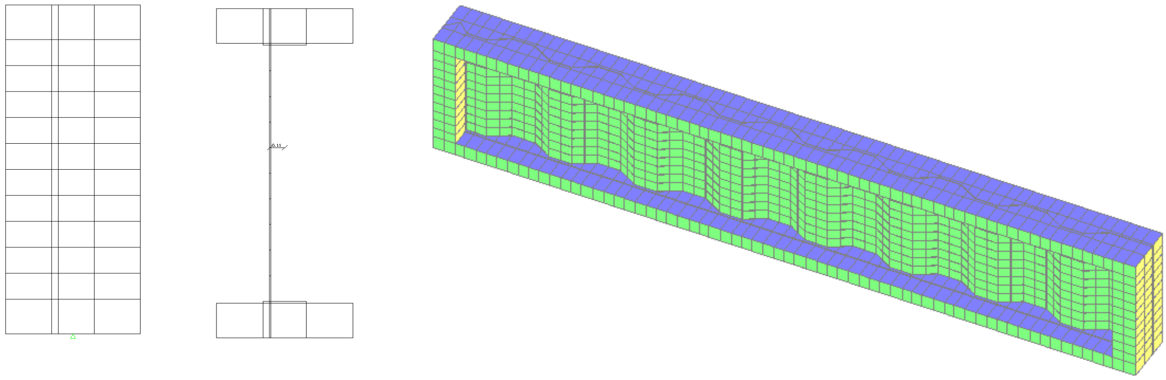

Fig. 5 *Cross-sections and calculation model of a girder with corrugated steel web*

Dependent on the effects to be investigated, the different structural parts can be defined either as composite beam elements or as finite elements. In this example, concrete and steel flanges have been simulated by beam elements, as distortion of these structural parts can be assumed negligible. The web is divided into 10 shell elements in order to allow for reproducing the accordion effect arising when the girder is subject to bending. Detailed comparisons of the results with previous approaches are work in progress and results will be presented later.

It should be noted, that the implemented types of Finite Elements use a variational approach with internal degrees of freedom in addition to the standard nodal DOF's of the 4 nodes of the shell, plate and wall elements or the 8 nodes of the volume elements. Therefore, the quality of these elements is equivalent to the quality of beam elements (3^{rd} order bending curve) and the compatibility between beam elements and continuum elements is fully guaranteed.

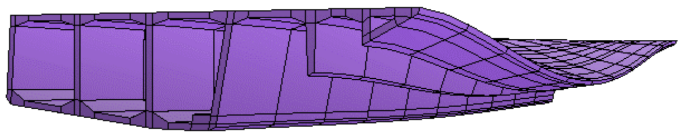

Fig. 6 *Deck behaviour for multi-cell sections*

In bridge deck analyses, where thin structural elements are predominant (e.g. slabs and webs of a hollow box section), a subdivision in thickness direction is normally not required and assumptions of shell theory can be used (no stress in thickness direction). Therefore, assigning the element type "Shell4" will mostly be appropriate. Volume elements are suitable for modelling solid cross-girders filling the whole hollow box. Plate and wall elements are special cases of the shell elements.

3.2 Degrees of Freedom for describing the deformability of the cross-section

The second step after creating the cross-section geometry is defining the degrees of freedom for describing the deformability of the cross-section as shown in Fig 6. Assigning so-called „Slave parts" to the cross-section or the cross-section parts respectively does this.

These "Slave parts" are sets of degrees of freedom related to a certain position in the cross-section. They represent the equations in the equation system being solved in the analysis.

All nodes of the finite elements will be automatically connected to the nearest "Slave part" with using the standard transformation of eccentric connections already described. This is in line with the general practice in RM software package, where the nodes do normally not coincide with the centre line of the elements, but have an eccentric position, mostly the centre point of the top slab.

3.3 Analysis

The final step in the input process is to set the switch FEM to yes for all segments where FE behaviour is wished or expected, and to do the export to the analysis part of the program. The finite element definitions provided by the user in the input part of the program automatically create so-called "Bodies" as objects in the database of the analysis program.

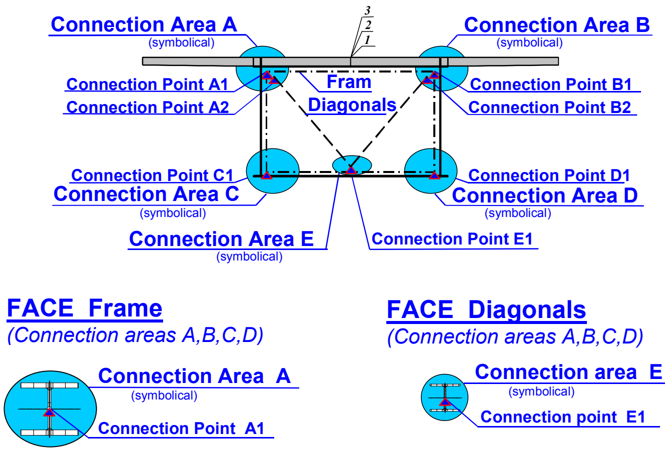

Fig. 7 *Introducing of internal constraints in body*

These "Bodies" are groups of FEM elements related to a common "father" (virtual beam element). For proper reference, the pre-processor creates body names identical to the names of the respective father elements.

Currently, Geometric Pre-processor automatically creates for all bodies integral results containing all elements of the body at the start point and end point of the father virtual beam element. I.e. integral results will represent the standard beam results. As shown in Figure 7, internal connection between different FE within the body will be introduced based on constrained degrees of freedom.

4. Practical examples

4.1 Typical Cases for the Use of Finite Elements

Bridge superstructures are very often beam-like structures, however, there are cases, where the deformability of the cross-section considerably influences the actual stressing state. Such situations are for instance:

- Parts of the cross-section shirk taking the nominal stresses due to warping effects. Known in this context is the shear lag effect referenced in 4.2, resulting in higher overall displacements due to additional flexibility and a raise of peak longitudinal stresses due to the decrease of the effective cross-section values.
- Local load application does not mobilize the whole cross-section due to the transversal deformability of the cross-section. This applies typically for wide, plate-like superstructures with multiple longitudinal beams, where eccentrically acting traffic loads fully hurt the directly affected girders. Creating a suitable equivalent girder grid model was required up to now to analyse these structures with sufficient accuracy. The new approach considerably eases the preparation of a suitable model as shown in chapter 4.3.
- Cases, where the overall behaviour is only marginally influenced, but local stresses essentially govern the bearing capacity or serviceability conditions. Examples are for instance local load transmission problems at abutments or clamped piers.

4.2 Shear Lag – a Typical Application

Shear lag effects are generally related to load transmission effects of high concentrated loads into the cross-section. Usually, these concentrated loads are support forces transmitted via shear stresses into the structure over a certain length. However, this effect is mostly not very important.

Shear lag as important phenomenon is mainly related to pre-stressed structures. Longitudinal stresses due to pre-stressing reach their nominal value sufficiently apart from the jacking points, where the stressing force is transmitted. Near to the stressed tendon compression stresses will be higher than the nominal stress based on plane-section assumption, and in the edge region the cross-section shirks the stressing. As most design codes require that no tension occur in the structure, calculating this compression stress reduction is essential to allow for a proper design.

Especially in major bridge projects with many tendons being stressed somewhere along the structure – as for instance in free cantilever bridges – this stress reduction also affects cross-sections near mid-span, which are design-relevant.

In beam model analyses, reducing the total cross-section area and moment of inertia in the respective regions and calculating with fictitious effective values is usually applied for approximately considering the shear lag effect. This reduction is based on rough approximations proposed in the design codes. In reality, the stiffness-relevant cross-section values depend very much from the actual loading. Using the Finite Element option effectively solves the problem as the deformability of the cross-section is fully taken into account.

4.3 Steel-Concrete Composite Bridges

A simple example of this type of bridges has already been presented in 3.1. A practical example of a more complicated composite bridge is shown below. The example is a rehabilitation project of a highway bridge in Austria. The bridge has been built in the middle of the 1970ies.

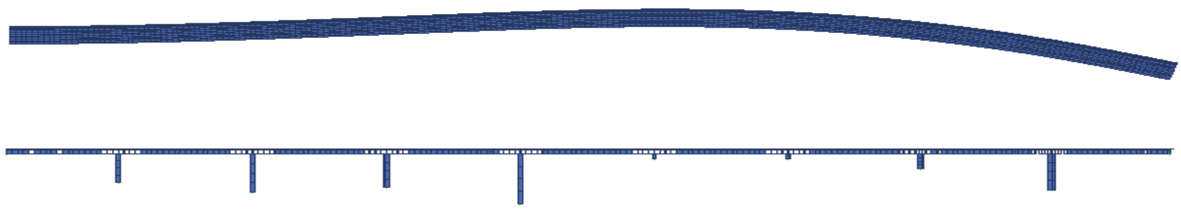

Fig. 8 *Composite bridge – horizontal alignment and elevation view*

A composite girder solution was chosen, as shown in Fig. 1, with two main steel girders and a cast-in-situ concrete deck. The horizontal wind truss is arranged under the slab and at the bottom of the steel section. The bridge is supported on eight piers with very different heights. In the horizontal plane the bridge is curved following the road alignment (Figure 8). Therefore, additional torsional effects and centrifugal forces due to live loads are present and 3D analysis and design is necessary.

Due to the increase of the traffic loading the bridge has been rehabilitated. Three rehabilitation steps have been provided. First, one side of the cantilever slab was extended. Second, additional steel members support the wider slab cantilever. Finally, the new layer of concrete is added to the slab and additional steel plates are added at the bottom of the steel girders.

It was necessary to provide an accurate analysis for the rehabilitation measures. Therefore the presented method has been chosen to verify the already existing grid model. The slab is modelled by using the virtual beam as substructure of shell elements with drilling DOF. In order to simplify the verification, similar virtual beams are chosen as in the grid model. Respective beam members model the steel girder and wind truss.

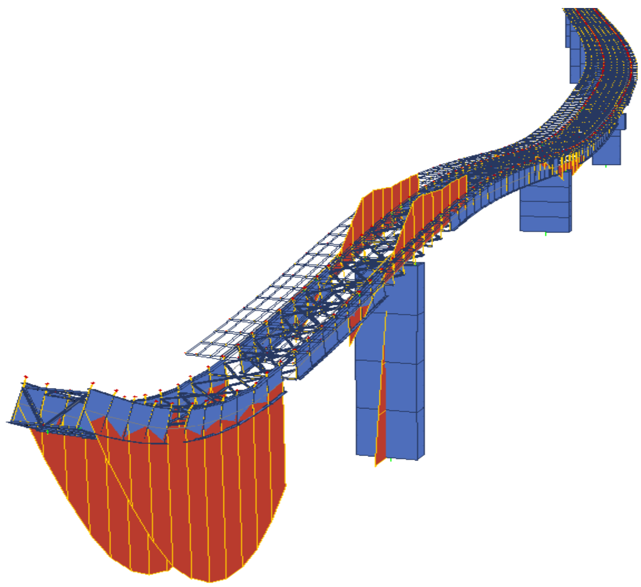

Fig. 9 *Composite bridge in construction stage - integral results in real and virtual beams*

The original construction of the bridge according to the knowledge of adopted construction sequence at that time was fully simulated with the new model in order to have a valid initial state for simulating the rehabilitation. Then, the rehabilitation steps are simulated, adding the widening of the concrete slab, steel support members for the cantilever, new concrete slab layer and additional steel section at the bottom.

Although a full Finite Element model would yield an increase of the equation system by at least one order of magnitude, the actual calculation time was only 30% higher than that of the corresponding grid model, due to reducing the number of global degrees of freedom in the hybrid model. The final hybrid model consists of 3375 nodes, 5170 virtual or real beams plus almost 8000 shell elements.

The comparison between FE model and grid model results showed good agreement with respect to the general behaviour of the bridge, longitudinal stresses and vertical displacements. Significant differences were detected with respect to transverse behaviour and torsional effects. The FE model also gives local effects in addition to global effects. Due to the flexibility of the cantilever slab, the supporting steel members get more compression stresses in FE model than in grid model.

It was in fact expected to get main differences between FE model and grid model results in effects governed by the shear stiffness of the slab, as it is well known that the grid model is not accurately modelling the behaviour of the structure in transverse direction. This has been impressively confirmed once more in this rehabilitation analysis.

5. Conclusion

A novel approach combining traditional structural beam analysis with finite element techniques in a hybrid model has been presented. This procedure allows for easily taking into account the deformability of the superstructure cross-section of bridges. Integrating the stress-distribution over the section – or arbitrary parts of the section - allows for applying standard proof checking routines based on beam results in the post-processing phase. An example of a rather complex composite bridge with steel cross-girders and bracings shows the suitability of the solution for practical applications.

The presented approach allows the engineer to introduce accuracy of analysis up to the desired level. The application example shows good agreement in longitudinal bridge elements. The expected differences in the transverse force distribution and torsional behaviour have been reconfirmed. The steel members supporting the cantilever slab show higher differences between the two models, where the grid model underestimates the compression stresses.

6. References

[1] T.BOGENSPERGER, „Erweiterte Stabtheorie und der gevoutete Träger im Brückenbau“,PhD TU-WIEN, August 2000

[2] O.C. Zienkiewicz: “The Finite Element Method”, 3rd ed., McGraw-Hill, 1977

[3] A. IBRAHIMBEGOVIC, R. L. TAYLOR, E. L. WILSON, “A Robust Quadrilateral Membrane Finite Element with Drilling Degrees of Freedom”, *International Journal for Numerical Methods in Engineering,* Vol. 30, pp. 445-457 (1990).

[4] L. DAMKILDE, M. GRONNE, “An Improved Triangular Element with Drilling Rotations“, *In Proceedings of NSCM 15, 15th Nordic Seminar on Computational Mechanics,* 18-19 October, 2002, Aalborg, Denmark, pp. 135-138.

[5] R. KOUHIA, R. STENBERG, “A Simple Linear Nonconforming Shell Element“, *In Proceedings of IASS-IACM 2000, Fourth International Colloquium on Computation of Shell & Spatial Structures,* June 4-7, 2000, Chania – Crete, Greece, pp. 1-10.

[6] C. A. FELIPPA, “An 8-Node ANS Solid-Shell Element”, *Report to Sandia National Laboratories, Albuquerque, NM,* July 28, 2001, pp. 1-19.

[7] T. J. R. HUGHES, T. E. TEZDUYAR, “Finite Elements Based Upon Mindlin Plate Theory With Particular Reference to the Four-Node Bilinear Isoparametric Element”, Journal of Applied Mechanics, September 1981, Vol. 48, pp. 587-596.

[8] SMITH L., *Limit Analysis and Concrete Plasticity*, Prentice Hall, New York, 2003, p. 360.

[9] K. DARILMAZ, N. KUMBASAR, “Noktah, Gerilme Secimli Bir Hibrid Mambran Sonlu Eleman”, *IMO Teknik Dergi,* 2005, pp. 3643-3655, Yazi 240.