

Design checks and nonlinear response of a full 3D model of a box girder bridge

Chantal Frissen

Engineering Consultant
TNO DIANA BV
Delft, the Netherlands

Chantal Frissen, born 1972, received her civil engineering degree from the Delft University of Technology, Department of Civil Engineering.

Ane de Boer

Civil Engineer
Ministry of Infrastructure and the Environment,
Utrecht, the Netherlands

Ane de Boer, born 1951, received his civil engineering and PhD degree from the Delft University of Technology.

Gerd-Jan Schreppers

Director TNO DIANA BV
TNO DIANA BV
Delft, the Netherlands

Gerd-Jan Schreppers, born 1963, received his mechanical engineering degree and PhD. from Eindhoven Technical University.

Summary

This paper presents a new approach for design and re-examination of a box girder bridge using a validated 3 dimensional finite element model. The Ministry of Infrastructure and the Environment in the Netherlands wants to quantify the remaining capacity of bridges as present traffic loads of bridges is higher than the loads for which bridges have been designed. Design checks are performed on a full 3-dimensional solid model and the same model is used to predict crack initiation, crack-development and yielding of steel reinforcements.

The analyses result in a prediction and quantification of the required amount of reinforcement, crack pattern, crack opening and plastic deformation of reinforcement, in addition to the traditional output of deformations, stresses and strains. The full 3D modelling approach makes it possible to predict the effect of refurbishing actions for repair and strengthening the bridge accurately. All the analyses are done using the finite element program DIANA version 9.4.4[1].

Keywords: box-girder bridge, post-tensioning, finite elements analysis, design and strengthening, 3D solid model

1. Introduction

Most of the bridges in the Netherlands have been built around 1970 and have not been designed for today's amount and heavy traffic. The Ministry of Infrastructure and the Environment in the Netherlands wants to know the remaining capacity of these bridges. There are many ways to re-examine structures. Where the remaining capacity is critical, a full 3D nonlinear detailed finite element analysis could be the choice. The box-girder bridge "Heteren" was analysed this way[2]. The model was based on the most realistic description of the bridge geometry and comprises an almost complete and detailed description of reinforcements, like we have today in CAD-systems. After some iterations of fine-tuning, analysis results in terms of crack patterns, crack widths and deformation showed good agreement with observations. This validated 3D model will be re-used to present a new approach in re-examining the reinforcement quantities of concrete structures and in a common nonlinear analysis.

First the re-design analysis is demonstrated using the new application DESIGN in DIANA. Here the required amount of reinforcement is checked for the new load configuration, five traffic lanes, while the bridge was originally designed for only two lanes plus one secondary lane.

The second analysis considers nonlinear material behaviour to calculate the crack width and plastic yielding of reinforcement. The entire workflow of these calculations is presented including the assessment of the bridge after strengthening.

2. Bridge Heteren

In this paper the Heteren bridge is defined by a validated 3D finite element model.

2.1 Geometry

The bridge is constructed as a 17 spans double box-girder bridge in the Netherlands with a total length of 986 meters. In the bridge there are two locations with rigid dowels (red arrows fig. 1a). During inspection, cracks were observed in the webs between the second dowel and the neighbouring pier 10 (fig. 1b). That is the reason why only five spans were modelled and analysed. The height of the box-girder is 2750 mm and the width is 16850 mm. The Heteren bridge is visualised in figure 1.

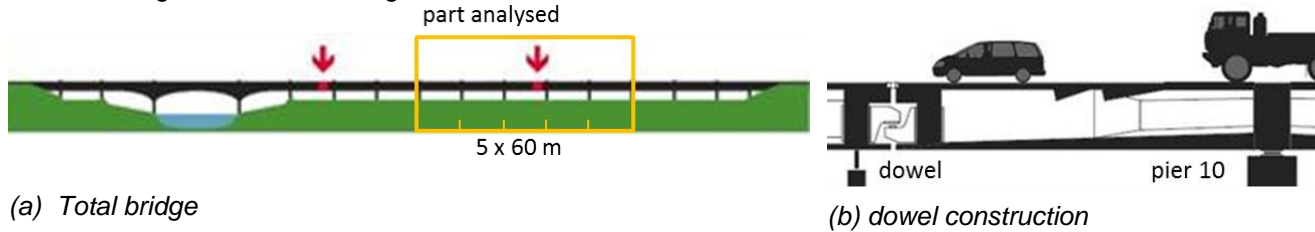


Fig 1: Bridge Heteren

2.2 Finite element model of the 5 spans

The 5 span structure is pre-stressed in longitudinal direction by 216 curved post-tensioned tendons in the webs. Also 22 additional straight post-tension tendons are present in longitudinal direction in the deck at pier 10. Furthermore, the deck has been post-tensioned in transversal direction using a tendon at every 500 mm. Finally, 72 tendons in transversal direction are modelled in each pier.

A 3-dimensional finite element model of solid elements was defined for the concrete body, taking into account construction details, such as variations of thickness of flanges and webs of the box, man-hole-openings and anchor-bolts. The finite element model consists of circa 240.000 lower order volume-elements, mainly of brick-type. Almost the complete and detailed composition of reinforcements has been modelled. In figure 2 only a part of the finite element model is visualised.

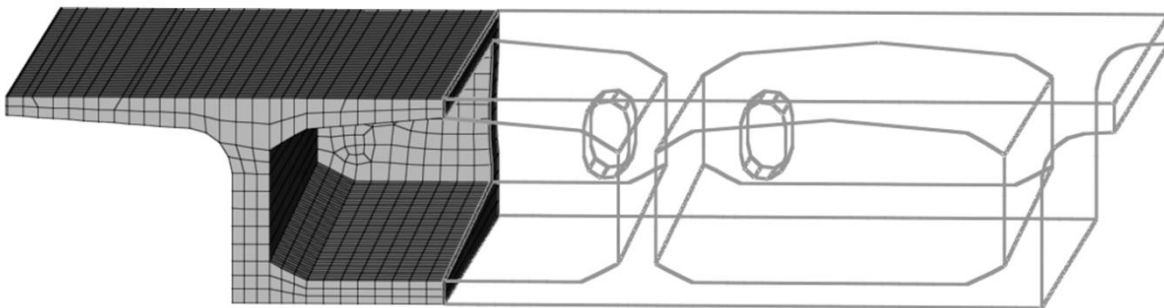


Fig. 2: Part of the finite element model Heteren bridge

The support blocks at the piers are modelled with interface elements. Each pier counts 2 support blocks almost near the outside webs. The mid web does not have a support block.

2.3 Material behaviour

In 2010 samples were taken out of the bridge to determine the existing material behaviour of the concrete, tendons and reinforcements. The results of these material tests are presented in the tables 1 to 3:

Table 1: Material properties Concrete (B55)

Parameter	Symbol	Value
Young's modulus	E	33077 N/mm ²
Poisson ratio	ν	0.15
Tensile strength	f_t	3.18 N/mm ²
Compressive strength	f_c	43.75 N/mm ²

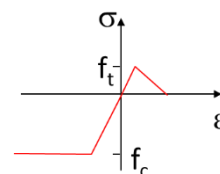


Table 2: Material properties Reinforcement (FeB 500)

Parameter	Symbol	Value
Young's modulus	E	200000 N/mm ²
Poisson ratio	ν	0.3
Yield stress	f_y	333 N/mm ²

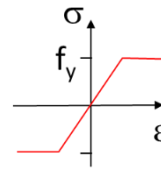
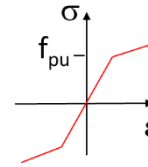


Table 3: Material properties Pre-stressed Tendons (FeP 1860)

Parameter	Symbol	Val
Young's modulus	E	200000 N/mm ²
Poisson ratio	ν	0.3
Yield stress	f_{pu}	1367 N/mm ²



2.4 Loads

The bridge was built in 1970 and designed for 2 traffic lanes and one extra separate lane. Meanwhile the number of lanes has been extended to 4 and today the bridge counts 5 lanes. The following loads are applied in both the design and the material failure analysis: dead weight, post tensioning, distributed load including crash barrier etc., asphalt load, 5 lanes distributed (Q-mobile) and an axle truck load (P-mobile), both according to the Eurocode1[3,4]. The truck load is located at the worst case location for shearing, which is between the dowel and pier 10.

3. Design analysis

A design analysis is performed on the bridge model loaded with the five traffic lanes (amongst the other loads) to check the required amount of reinforcement in ultimate limit state. The required amount of reinforcement is calculated and compared with the existing (applied) area of reinforcement. The new application in DIANA, called DESIGN, will be used for this analysis.

3.1 Background theory

The calculation of the required area of reinforcement is based on the prediction of reinforcement forces and moments in a linear elastic analysis. The calculation is based on the assumption that only the reinforcement resists the tension forces, the concrete only the compression forces. The concept is also based on the fact that the reinforcement in both directions are located in one upper and one lower plane of the structure. The location of these design-reinforcement grids can be defined by the user.

The reinforcement loads are expressed by the combination of membrane forces and bending moments. The Cauchy stresses in the solid elements should be integrated along a line normal to a reference surface to get these distributed forces and moments. This will be done automatically when using so called "composed surface elements" in DIANA.

These elements should be generated as being the reference plane of the structure (or part of it). The advantage of using these elements is that they can be modelled independently of the volume mesh. No connection with the structural mesh is required. These elements are only used for post processing purpose to get distributed forces and moments and they will not contribute to the stiffness of the structure.

3.2 Approach Heteren bridge

The amount of reinforcement is checked at the location where cracks were observed. The cracks are located in the webs between the dowel and pier 10. For the design analysis, composed surfaces are defined at the centre planes at the three webs representing the reference planes for the integration of the Cauchy stress over the thickness of the webs into distributed forces and moments. In addition two design grids are modelled in every web, one at each side of each web. This is visualised in figure 3.

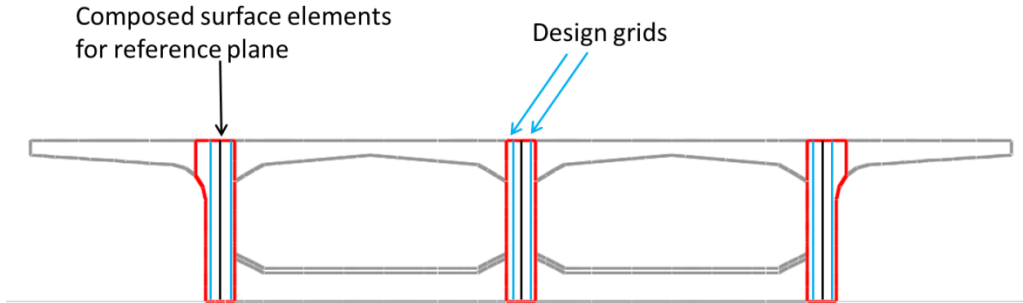
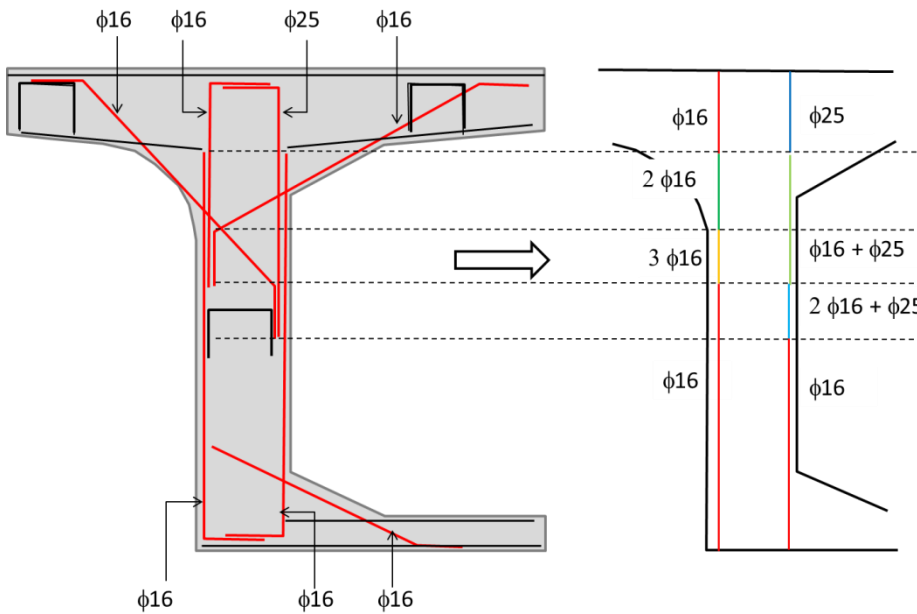


Fig. 3: Cross section view of the webs with the reference planes and reinforcement grids

The required amount of reinforcement is calculated and compared with the applied amount of reinforcement. All existing reinforcement in the webs must be converted to just one reinforcement grid at each side of each web. The applied reinforcement area can vary within this plane of reinforcement. In figure 4a, the reinforcements in the left side web are visualised. Only the "red" reinforcements are considered for the design-reinforcement in vertical direction. All reinforcements are placed 200 mm centre to centre. The translation of real-reinforcements to the two design reinforcement grids in this web is shown in figure 4b. The reinforcement in the mid web on both sides is equal to the reinforcement at the outside of the left web. The same is done for the reinforcements in horizontal direction. For practical reasons the same location of reinforcement grid is used for both the reinforcement in horizontal and vertical direction.



(a) Existing reinforcement in left side web
Fig 4: Reinforcement in vertical direction

(b) Applied reinforcement in Design analysis

3.3 Results

With the design analysis the required amount of reinforcement is calculated. The required amount should be larger than the applied area of reinforcement. That is why in the Design application of DIANA also the ratio between the required and applied amount of reinforcement is available as a calculated result. This ratio value must be equal or less than 1.0 in order to comply with the Eurocode2[5]. In figure 5a the required amount of reinforcement in vertical direction is given in the six design reinforcement grids. Here you can see that the maximum amount of reinforcement required in vertical direction for the bridge with the new 5 traffic lanes is $5.6 \text{ mm}^2/\text{mm}$. This is larger than the maximum area applied of $4.47 \text{ mm}^2/\text{mm}$ ($2 \times 16 + 1 \times 25$) which is given in figure 4c. The ratio between the required and applied area of reinforcement (ASRAT) is given in figure 5d. At some locations, the value of ASRAT is larger than 1.0. This means that the bridge with the modelled reinforcement, having five traffic lanes, does not comply with the Eurocode2. After discussing this result with Ministry of Infrastructure and the Environment it was concluded that there is more basic reinforcement in the bridge than was assumed in this 3D model (see also figure 4). The amount of reinforcement in the model is updated in a second round of analyses. Also, using a yield-stress of 435 N/mm^2 given by the Eurocode2, instead of the 333 N/mm^2 , affects the design in a positive way. Despite the reinforcements in the bridge not being complete, it was demonstrated that with the design analysis, the problem areas can be identified and correspond with observations.

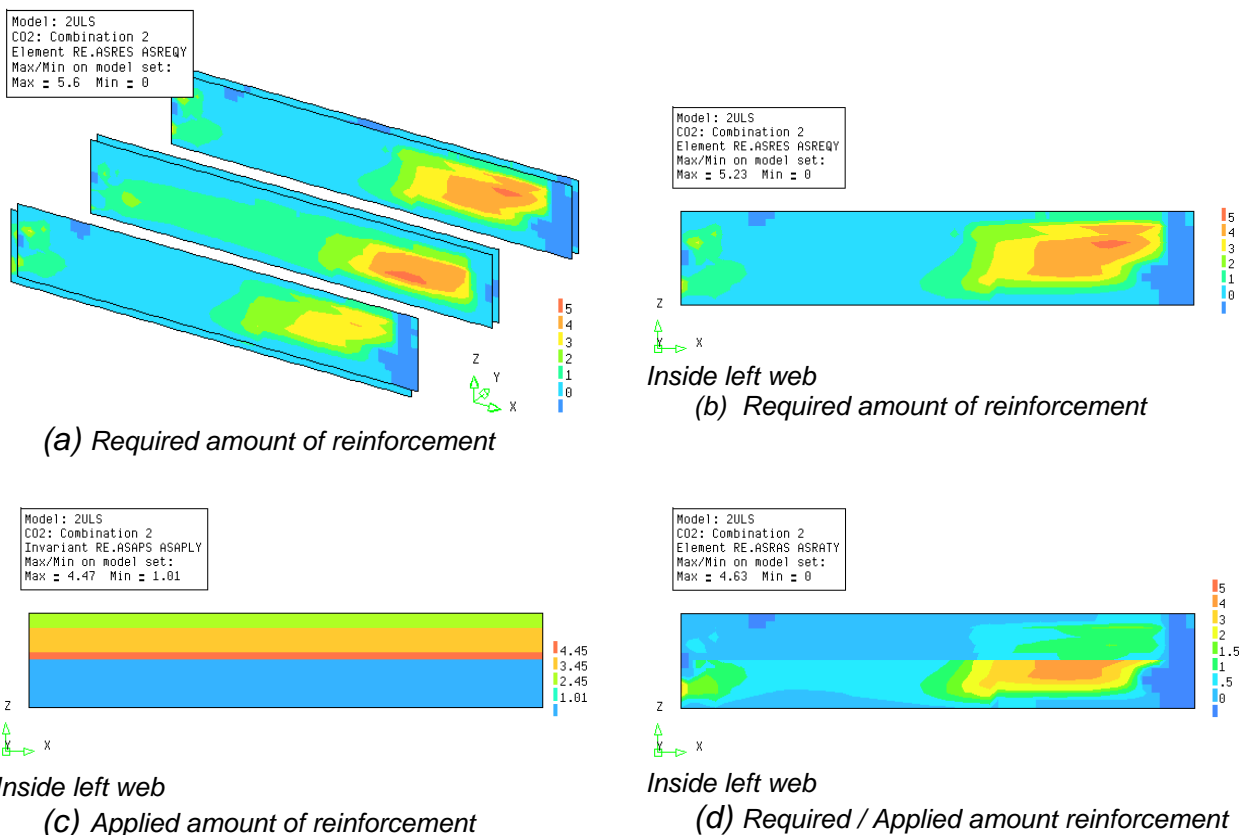


Fig. 5: Design results: Amount of reinforcement [mm^2/mm]

4. Stiffness adaptation analysis

Checking the area of reinforcement is just one of the checks in the design stage of a structure, or in this case in re-examining the bridge. For a damaged structure also the crack width in Serviceability Limit State should be checked. This crack width can be efficiently predicted with the new stiffness adaptation (STADAP) application in DIANA. This recently developed application reduces the total elapse time in comparison with a full nonlinear analysis enormously.

4.1 Background theory

With a stiffness adaptation analysis results like crack patterns, crack openings and yielding of reinforcement can be predicted without running a full nonlinear analysis. This type of analysis performs a sequence of linear static analyses. In a subsequent iteration the elastic stiffness is reduced in those integration points in which the stresses in a previous iteration were beyond a user-specified uni-axial stress-strain curve. The stiffness is reduced in the direction of the maximum stress, such that, with the same strain, the maximum stress is mapped on the stress-strain curve. In this case the isotropic elastic stiffness is changed into an orthotropic elastic stiffness with a reduced Young-modulus in the direction where principal stress exceeds the stress-strain-curve. The stiffness is evaluated and modified in every integration point and in every principal stress direction. This concept is visualised in Figure 6.

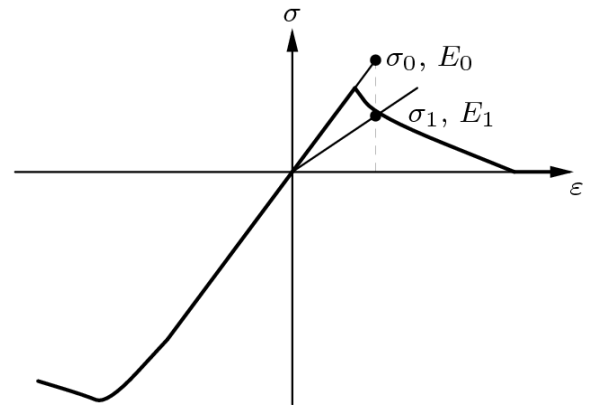


Fig. 6: Uni-axial stress-strain curve with stress and stiffness reduction

4.2 Approach Heteren bridge

The stiffness adaptation analysis is performed using the earlier described 3D volume model of the bridge. The new 5 lanes traffic load is used to calculate the maximum crack opening in serviceability limit state. For this STADAP calculation a nonlinear uni-axial stress-strain curve has only been applied to the concrete of the 3 webs between the joint and pier 10. This nonlinear curve has been given in table 1. The rest of the 3D model was kept linear elastic.

Also the reinforcements in these webs must be modelled to get a more realistic nonlinear behaviour. So the design-reinforcement grids are removed and replaced by almost all existing reinforcements. For the reinforcements in span-directions, embedded reinforcement bars are used. The reinforcements in vertical and transversal direction are modelled with reinforcement grids. A nonlinear uni-axial stress curve, as shown in table 2, is applied to these reinforcements.

4.3 Results

The crack widths are calculated in serviceability limit state using the STADAP analysis. This result is visualized in Figure 8. This figure shows that cracks occur in all the three webs. The maximum crack opening equals 2.68 mm and occurs in the right-hand side web where the highest lane load is positioned. This maximum crack width is larger than the crack-width that is allowed by Eurocode2.

In a STADAP analysis two stiffness values are used: one for the tensile regime in the direction of the tensile principal strain, and one for the compressive regime in the direction of the compressive principal strain. The results are presented as reduction factors in the tensile and compressive directions. If the reduction factor is equal to 1.0, the stiffness is equal to the original Young's modulus. The minimum stiffness reduction factor is set to 0.001. The tension reduction factor is given in figure 9. This figure shows that the lowest tension reduction factor equals 0.001 (which is the lowest remaining stiffness possible). Comparing with figure 8, one can see that the location of the cracks corresponds with the area of the lowest tension reduction factors.

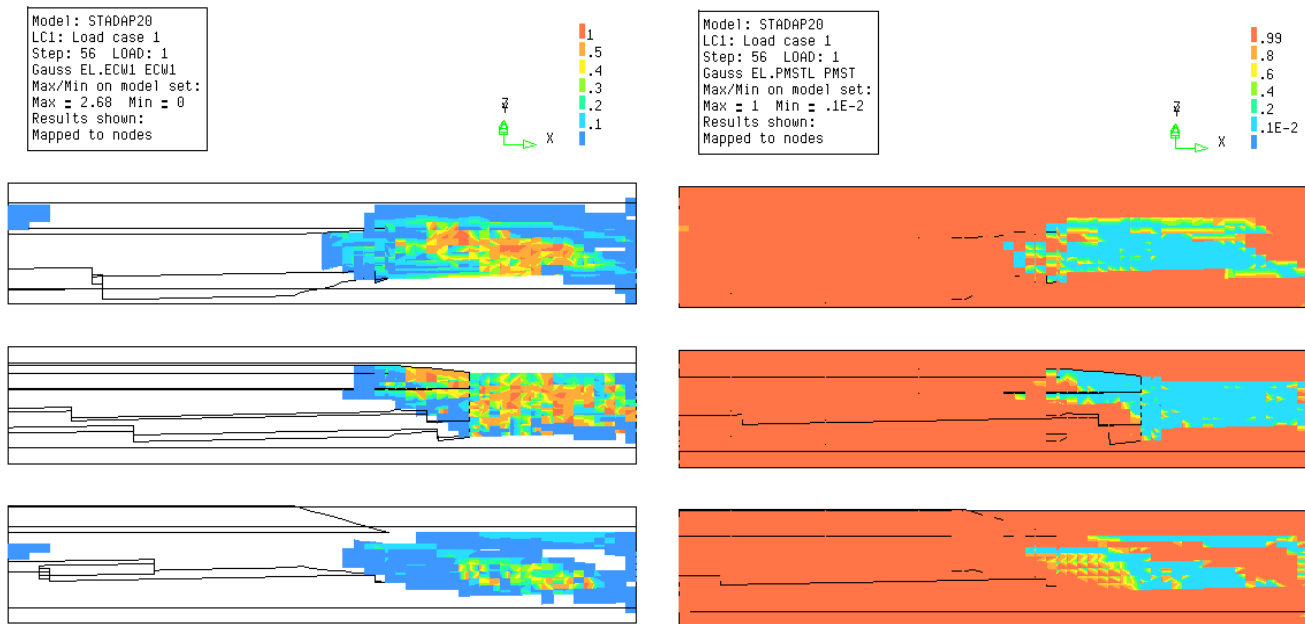


Fig. 8: Crack opening at Serviceability Limit State [mm] Fig 9: Tension reduction factor in the 3 webs [-]

Stiffness reduction results in lower (remaining) stresses in the cracked concrete. The loading is taken over by the reinforcement. The tension reduction factor of the reinforcement grids indicates if the reinforcement grids are yielding or not. The tension reduction factor in the vertical reinforcement grids are all equal to 1.0, so the reinforcement grids are not yet yielding in the serviceability limit state.

5. After strengthening the bridge

After cracks were observed during inspection in the bridge, several numerical analyses have been performed to calculate the remaining strength of the bridge. Consequently the bridge has been strengthened in 2010. At pier 10 extra curved post-tensioned tendons are applied in the webs for additional pre-stressing the webs at the location where the cracks are observed. The remaining strength of the bridge after strengthening has been checked with numerical analyses. For modelling the strengthening only additional post-tensioned tendons are added to the finite element model and both the design and stiffness adaptation calculations can be rerun. Modelling extra tendons in an existing finite element model in DIANA is easily done because of the use of embedded reinforcements. These reinforcements can be modelled in DIANA independently from the structural solid element mesh.

6. Conclusions

This paper showed a new approach for designing and re-assessment of structures using a 3D finite element analysis. It has been demonstrated that it is now possible to do the following checks and analyses relatively easy and fast:

- ▶ Amount, layout and specifications of reinforcements can be checked against design-codes for different loading conditions for the undamaged bridge.
- ▶ The development of crack-patterns and crack-widths in the bridge can be predicted and quantified at different loadings by using a stiffness adaptation analysis.
- ▶ For a damaged bridge, the effect of the reduced stiffness, due to cracking, on the nonlinear behavior of the reinforcement (yielding) can be checked.
- ▶ The effect of refurbishing actions to repair or strengthen the bridge can easily be assessed.

For these checks and analysis on a 3D model, the new applications DESIGN and STADAP (stiffness adaptation analysis) in DIANA have been used in combination with composed surface elements to integrate the Cauchy stresses into distributed forces and moments. These analyses can be a good alternative for a complex full nonlinear analysis, because they are easy to use and relatively fast. All the results, like required amount of reinforcement, crack pattern and crack opening and plastic deformation of reinforcements can be predicted and quantified, in addition to the standard results of deformations, stresses and strains.

Today, the design of new structures is mainly done with dedicated design software programs which are based on relatively simple finite-element technology using plate and beam elements. These models are not suitable for accurate prediction of crack-widths and plastic yielding of reinforcement. The availability of full 3D solid models, which are validated during life-time, offers important advantages for conscience and efficient life-time management of structures.

7. References

- [1] JONNA MANIE, DIANA User's Manual, Release 9.4.4, 2011 (www.tnodiana.com)
- [2] NIELS WILLEM KOSTENSE., Feasibility of a full concrete viaduct modeled with volume elements, Part I & II, Msc Thesis, Delft University of Technology, Delft, August 2010
- [3] CEN2002, EN 1991-1-1 Eurocode 1: Actions on structures - Part 1-1: General actions - Densities, self-weight, imposed loads for buildings
- [4] CEN2003, EN 1991-2 Eurocode 1: Actions on structures - Part 2: Traffic loads on bridges, 2003, Paris
- [5] CEN2004, EN 1992-1-1 Eurocode 2, Design of concrete structures, European Committee for Standardization, 2004, Paris