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
c.frissen@tnodiana.com

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

Ane DE BOER
Civil Engineer
Ministry of Infrastructure and the Environment,
Utrecht, the Netherlands
ane.de.boer@rws.nl

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
g.schreppers@tnodiana.com

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 box-girder bridge using a validated 3 dimensional finite element model. The Ministry of Public Works in the Netherlands wants to quantify the remaining capacity of the concrete bridges because these bridges were not designed for the amount and heavy traffic today. Design checks are performed on a full 3-dimensional model with volume elements, and the same model is used to consider non-linear material behaviour. 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. With this approach also the effect of refurbishing actions for repair and strengthening the bridge can be predicted 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 volume model

1. Introduction

In this paper 5 spans of the Heteren bridge, a post-tensioned double box-girder bridge constructed in 1972 in the Netherlands, are defined by a validated 3D finite element model [2]. The model consists of circa 240.000 lower order volume-elements and was based on the most realistic description of the bridge geometry and comprises an almost complete and detailed description of reinforcements. A part of this finite element model is shown in Figure 1. The model is loaded with: dead weight, post tensioning, distributed load, asphalt load, 5 lanes distributed (Q-mobile) and an axle truck load (P-mobile), both according to Eurocode1.

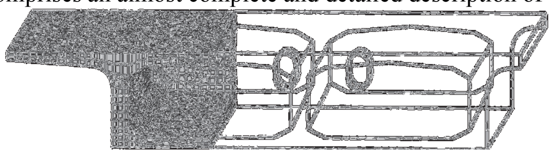

Fig. 1: Small part of the finite element model

2. Design analysis

A design analysis is performed on the bridge, loaded with 5 traffic lanes (Eurocode2) instead of the 3 lanes at the design stage, to check the required amount of reinforcement in ultimate limit state. Cracks were already observed during inspections in the webs. The required amount of reinforcement is calculated in the 3 webs. In each web 2 reinforcement grids are defined, one at each side of each web. The results are visualised in figure 2. Modelling the basic reinforcement in the bridge, it was demonstrated that with the design analysis software application, the problem areas can be identified and correspond with the damaged observations.

3. Fast stiffness adaptation analysis

The crack width can be efficiently predicted with the new Stiffness adaptation (STADAP) application in DIANA. This type of analysis performs a sequence of linear static analyses in which the elastic stiffness is reduced to simulate cracking considering non-linear material behaviour.

https://doi.org/10.2749/222137813806521351

This application reduces the total elapse time in comparison with a full nonlinear analysis enormously. The crack opening is visualized in Figure 3. This figure shows that cracks occur in all the three webs which are larger than the crack width that is permitted according to Eurocode 2[3].

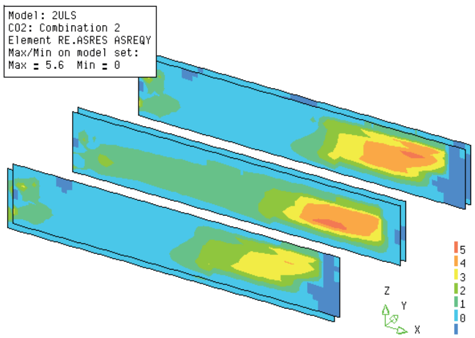

Fig 2: Required amount of reinforcement in Ultimate limit state [mm²/mm]

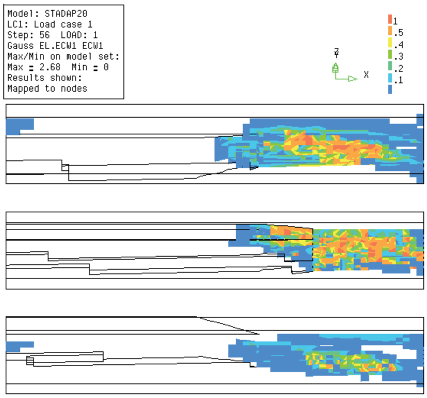

Fig 3: Crack opening in the webs at serviceability limit state [mm]

4. 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.

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, shell and beam elements. These models are not suitable for accurate prediction of shear, crack widths and plastic yielding of reinforcement. The availability of full 3D models, which are validated during life-time, offers important advantages for conscience and efficient life-time management of structures.

5. 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] CEN2004, EN 1992-1-1 Eurocode 2, Design of concrete structures, European Committee for Standardization, 2004, Paris