Experimental investigation on the bridge segments with transversally curved bottom flange

Filip Ljubinković¹, João Pedro S C Martins¹, Helena M S Gervásio¹ and Luís A P Simões da Silva¹

¹ISISE, Department of Civil Engineering, University of Coimbra, P-3004 516, Coimbra, Portugal

E-mail: filj@uc.pt

Abstract. The main objective of this research project is to develop solid knowledge on the structural behavior of curved steel panels for optimized applications in steel and composite bridges. Moreover, the goal is to extend the EN 1993-1-5 design methodology for transversally curved panels subjected to various in-plane load cases. For that purpose, two identical prototype box-girder bridge segments in scale 1:3, with the transversally curved bottom flange and with two different steel grades (S460 and S690) are to be tested. Bridge segments are tested as three-point bending tests, simulating the bridge behavior near the intermediate support, where the interaction between bending moment and shear force occurs in the lower curved flange. This paper is mainly focused on the preparation process prior to the experiments and the description of the test layout. The first specimen was tested recently and thus, only the preliminary results are presented. Later on, based on these results, a numerical model will be validated and used for a parametric study, where the influence of several parameters on the behavior of these innovative bridge prototypes will be investigated.

1. Introduction

In recent years, the use of the cylindrically curved panels as a bottom flange in steel bridge applications has experienced an uprising trend. One typical example of the box-girder cable-stayed bridge with the curved bottom flange is given in figure 1.

Figure 1. Ebro River Pedestrian Bridge, Zaragoza, Spain.

Figure 2. The geometry of a cylindrically curved panel.
The main parameters required to define the geometry of cylindrically curved panels are schematically presented in figure 2, where \(a\) and \(b\) are the length and width of the panel, \(a_{loc}\) and \(b_{loc}\) are the length and width of the corresponding subpanel, \(t\) and \(R\) are the thickness and radius of the panel, and finally, \(\varphi\) is the sectorial angle of the panel. The aspect ratio \(\alpha\) is defined as the ratio between the length and width of the panel or sub-panel, whereas the curvature parameter \(Z\) is defined by equation (1)

\[
Z = \frac{b^2}{R \cdot t}
\]

An investigation carried out by Reis et al. [1] shows that as many as 18 bridges with curved cross-sectional parts have been successfully designed so far. The ranges of curvature parameters commonly used in these 18 applications are given in figure 3. Moreover, the global aspect ratio usually varies between 0.5 and 1, whereas the local aspect ratio varies between 3 and 6, as cited in Martins et al. [2].

![Figure 3. Parameters range (a) Z_{global} - whole bottom flange and (b) Z_{local} - sub-panels between longitudinal stiffeners.](image-url)

However, the existing design standards for plated structures (i.e. EN 1993-1-5 [3] and EN 1993-1-6 [4]) still do not provide the design rules for this kind of solutions, such as bridges with a transversally curved bottom flange. The designers are, therefore, mainly relying on their own experience and on the use of the advanced Finite Element Analysis. Therefore, a huge step forward in the understanding of the structural behavior of bridges with the curved panels is urgently required. This study aims to develop a clear design approach for curved panels, using experimental and numerical approach.

Therefore, two identical bridge prototypes, with a scale of 1:3 with respect to a real study case of a bridge with cylindrically curved bottom flange, are subjected to experimental investigation. The geometrical parameters (i.e. curvature parameter and aspect ratio) of the bridge prototype are carefully chosen to fall within the common ranges of these parameters. The exact shape and dimensions of the specimen will be described in the following chapters.

2. Predesign of the test specimen

2.1. General

The goal of this part of the study was to prepare and adapt the bridge prototype segments to fit the capacities of the laboratory at the University of Coimbra, in terms of geometry and load application. The whole process from the initial case study of the real bridge, designed by GRID International, until the final adoption of the geometry for the future test specimens is explained.

2.2. Case study

The studied structure is a continuous five-span composite steel-concrete girder bridge with standard abutments sitting on embankments. It is deemed to carry a roadway corridor with a single carriageway.
The superstructure comprises two lateral spans of 37.8 m and three central spans of 54 m, in a total of 237.6 m length (figure 4).

This cross-section shape was actually proposed as a variant of an already built bridge, designed by GRID International. The alternative solution with cylindrically curved steel panels has been developed adopting a box girder section, with a constant depth of 2.25 m. The cross-section consists of a steel girder and a concrete slab supporting the roadway. The bridge deck is 10.93 m wide, carries two traffic lanes 3.70 m wide and two pedestrian walkways 1.6 m wide in each direction.

Figure 4. Elevation span layout.

The box girder is formed by: (i) a curved bottom panel, 3.4 m wide, with a radius of 2.5 m and longitudinally stiffeners (0.25x0.025 m flats each 0.95 m), (ii) webs with 46 degrees of inclination with regards to the horizontal, (iii) and standard plate top flanges, 0.8 m wide and 0.08 m thick (figure 5 and figure 6). The reinforced concrete slab is supported by cross-beams at each 2.16 m. Over the supports, a 0.03 m thick closed diaphragm was adopted. The web thickness has the same thickness and steel distribution as the cylindrical panel, and it is longitudinally unstiffened.

Figure 5. Span cross-section. Figure 6. Pier cross-section.

The main geometrical parameters (i.e. global and local curvature parameters, aspect ratios and plate slenderness) of the proposed solution are given in table 1. It is worth noting that the global curvature, $Z_G = 235$, and the local curvature, $Z_L = 12$, are in the upper bound of existing bridge solutions (see figure 3) in order to perceive the benefits of adopting higher curvatures. Additionally, regarding the aspect ratio, sub-panels between stiffeners are classified as long panels, $\alpha_L = 2.3 > 2$, whereas the whole stiffened plate as short panel, $\alpha_G = 0.5 < 1$.

| Case study | $Z_G$ | $\alpha_G$ | $b_G/t$ | $Z_L$ | $\alpha_L$ | $b_L/t$ |
|------------|-------|------------|---------|-------|------------|--------|
| Existing data (avg.) | 235 | 0.5 | 140 | 12 | 2.3 | 32 |
| Case study | 100 | 0.7 | 165 | 3 | 4 | 28 |

2.3. Geometry of the test specimen

The experimental layout aims to imitate the behavior of the mid support of the previously defined case study, where the length of approximately 0.2xL ($L \approx 54$ m) on both sides of the mid support (i.e. 0.4xL in total) corresponds to the distance between two zero-bending moments. This leads to the segment with
the total length of approximately 26 m. This segment is proportionally scaled down 3 times in order to fit the laboratory capacities in terms of dimensions and loading applications. Additionally, several simplifications were adopted:

- The cross-section is considered as a homogeneous steel section, and not as a composite one, where the reinforcement steel above the mid-support is replaced by the corresponding steel plate, resulting in a top flange with the constant thickness
- The inclination of the slab of 2.5% is neglected, leading to a flat top flange
- The cross-section of the bridge which participates in load-bearing capacity is consisted only of the box, while the cantilevers that carry the load from the footways are omitted
- After the scaling down 3 times, all the dimensions are rounded to a smaller whole number in order to ease the fabrication process

Finally, the exact dimensions of all the relevant parts of the specimens are given in the in figure 7 and figure 8, whereas table 2 lists all the dimensions which may not be seen in these figures.

![Figure 7](image1.png) Adopted cross-section geometry.  
![Figure 8](image2.png) Geometry of the bottom flange.

| [mm] | Symbol | R = 1:1 | R = 1:3 |
|------|--------|--------|--------|
| Total length | $L$ | 25920 | 8640 |
| Top flange thickness | $t_f$ | 80 | 25 |
| Length (segment) | $a$ | 2160 | 720 |
| Stiffener thickness | $t_{st}$ | 25 | 8 |
| Cross-frame thickness | $t_T$ | 16 | 5 |
| Diaphragm thickness | $t_d$ | 30 | 10 |

2.4. Preliminary numerical study

For the adopted geometry of the specimen from the previous step, the final step before the definition of the test setup was a preliminary numerical simulation of the test, based on the FE method. The arc-length Riks’ [5] method was used to check what is the ultimate load that the bridge specimen may sustain and also, to predict the possible failure mode. This would be useful information for the instrumentation of the specimen which is described in the following section.

As for the material, the mechanical properties of the steel S460 and S690 are taken with the nominal values, increased by 15% and 10%, respectively, accounting for the hardening of the material ($E = 200$ GPa; $\nu = 0.3$; $f_{y,S460} = 530$ MPa and $f_{y,S690} = 760$ MPa). The real material properties will be tested subsequently after the test, and this law will be used for the final numerical model. Regarding the mesh, four-node shell elements with the reduced integration were used (S4R), with the mesh size of 150x150 mm$^2$, which is obtained after the mesh sensitivity analysis as a size that reduces CPU time and gives sufficiently correct results.

The load and boundary conditions are modeled in such a way to simulate as close as possible the future conditions of the experiment. They are schematically illustrated in figure 9. The concentrated
load, which is distributed by means of a rigid loading beam, is placed on one side of the specimen, at the position of the hydraulic jack. Additionally, the self-weight of the specimen is also introduced into the model. Regarding the boundary conditions, all three translations (in $x$, $y$, and $z$-direction) are prevented at the middle support, whereas on the opposite side of the load, 4 DYWIDAG cables are modeled with the appropriate axial stiffened ($k$), with Young's modulus equal to 205 GPa.

Finally, the geometric imperfections were also accounted for with the shape affine to the first Eigenmode, obtained by Linear Bifurcation Analysis (LBA) and with the amplitude equal to the $b_{loc}/200$, which is actually suggested in Annex C of EN 1993-1-5 for the flat plates.

The failure mode of the bridge specimen obtained numerically is presented in figure 10. As it is expected and intended, the failure occurs in the bottom (curved) flange, where both the longitudinal stiffeners and curved panel in-between buckled. The hypothetically maximum possible load of the test is equal to 2250 kN (i.e. 225 tonnes), which is obtained in case of the stronger (S690) specimen without imperfection. Based on these results, the drawings of the specimens are prepared and the specimens fabricated by MCE GmbH.

3. Test layout

3.1. General

For the experimental campaign, based on the previous results, the following material was designed and fabricated (figure 11):

- 1 bridge specimen made of steel S460 and another identical of S690
- 2 supporting feet where the mid support will be placed
- 1 loading beam (stiffened HEB300 profile)
- 2 supporting beams (back-to-back U260 profiles) to anchor 4 DYWIDAG cables
- 2 supporting beams (HEB300 profiles), to support two beams (back-to-back U260 profiles)
- 4 DYWIDAG prestressing systems, with the diameter of 47 mm, and length of 4.5 m

Except for the bridge prototypes, all the other auxiliary elements are made of steel S355. The elements were manufactured by MCE GmbH and delivered to the laboratory of the University of Coimbra, within the on-going research project OUTBURST (ref.: RFCS-2015-709782).

The bridge segment is loaded by a concentrated force on one side using the 600-ton hydraulic jack RCS-1002 (figure 12). The load is transferred then to the bridge over a load beam made of a stiffened HEB300 profile. The middle support of the segment is supported by two so-called support feet, designed particularly to sustain the huge axial force, which are connected between themselves by means of bolts. Finally, on the other side of the bridge segment, 4 DYWIDAG cables that are anchored to the ground, act as a counter-reaction, preventing the rotation of the bridge segment upwards. The cables are
supported by two simply supported beams, each of them made of two back-to-back UPN260 profiles, with sufficient space in-between for the cables to pass. The webs of the beams are well stiffened to sustain the huge local transversal forces which appear during the test. These two beams are supported by another two beams also made of HEB300 profile, which are connected by means of welded plates in the level of top and bottom flange (figure 13).

![Test setup](image1)

**Figure 11.** Test setup

![Hydraulic jack and loading beam](image2)

**Figure 12.** Hydraulic jack and loading beam.

![End support](image3)

**Figure 13.** End support – two 2xU260 beams (white) and two HEB300 beams (blue).

3.2. Instrumentation of the specimens

The data that are measured directly from the test are listed here:

- Externally applied force and reactions in the mid and end-support (with load cells)
- Vertical, lateral and longitudinal displacement of the bridge (with LVDTs)
- Strains distribution in the zone where failure is expected to occur (with strain gauges – SG)

3.2.1. Linear displacement transducers (LVDT). In total 10 LVDT TML are used to measure displacements in various positions along the segment. Namely, 7 of them measure vertical displacements, where two out of them (V-6 and V-7) are placed on the ends of the loading beam (see figure 12), 2 measure the longitudinal displacement (L-1 and L-2) at two ends of the specimen, and 1 measures possible horizontal (lateral) displacements (H-1) (figure 14).

![Position of the LVDTs](image4)

**Figure 14.** Position of the LVDTs.
3.2.2. Strain gauges (SG). For each of two specimens, it is planned to use 103 FLA-6-11 SG positioned in 3 different planes, namely 150 mm, 300 mm and 570 mm distant from the middle support, where the failure is expected. The SG are organized in such a way to measure both longitudinal and transversal strains, according to figure 15. To enable the installation of measuring equipment inside the section, two “windows” were left open and then closed (welded) afterward (figure 16). The cables from the inside of the bridge segment were led outside through one of the wholes (200x200 mm$^2$) deliberately left on the top flange for this purpose.

![Figure 15. Position of SG.](image1)

![Figure 16. SG on the welded “windows”.](image2)

3.2.3. Load cells. To measure the value of the load during the test, 7 load cells of the type C6A were used. Four of them (D1, D2, D3 and D4), with the capacity of 2MN were placed at the position of 4 DYWIDAGS, whereas the other three, with the capacity of 5MN, were placed at the position of the loading point (F) and also below the middle support (R1 and R2), according to the figure 17.

![Figure 17. Position of the load cells.](image3)

![Figure 18. Failure of the curved panel.](image4)

4. Test results

The first specimen (S460) was tested only recently and thus, in this document only the preliminary results are presented. The test was displacement controlled, with the constant speed of 0.01 mm/s until the collapse. The load application was stopped when the vertical displacement at the position of the load reached the value of approximately 117 mm, which occurred in the descending part of the force-displacement curve. The location and the pattern of the failure mode completely coincide with the ones obtained in the preliminary numerical analysis (see figure 10), with the obvious buckling of the curved panel in the bottom flange close to the mid-support (figure 18).

The maximum load reached a value of 1580 kN, which is practically the same ($\Delta \approx 1.5\%$) as the one obtained in the numerical analysis (i.e. 1602 kN), as it is presented in figure 19. The data recorded by four load cells at the position of the DYWIDAG cables is presented in figure 20. The sum of these four values at the moment when the ultimate load is reached is almost equal to the applied load ($D_{tot} \approx 1400$ kN) and also almost the same values were obtained in numerical analysis ($D_{num} \approx 1350$ kN), which confirms that the bridge statically behaved in the desired way.

Based on the vertical displacements at the ultimate load measured by LVDTs, the deformed shape of the segment is schematically illustrated in figure 21. The maximum longitudinal and lateral movements reached the values of 1.5 mm and 0.7 mm, which may be considered negligible.
Concluding marks and future work

Based on these preliminary results of the first test, it was concluded that the experiment was successful and in accordance with the targeted behavior, which gives a certain confidence ahead of the second test. The minor adjustments will be subsequently performed in the numerical model, namely, the real material law will be incorporated, as well as the appropriate geometric and material imperfections (i.e. residual stresses). Also, a study on the influence of the axial stiffness of the DYWIDAG cables will be performed, since the one in the numerical model led to a slightly more rigid system, resulting in a smaller vertical displacement (e.g. 90 mm at the point of the load). Finally, it is necessary to investigate deeply the stress-strain relations in the bottom flange close to the mid-support, where the interaction between shear and compressive stresses occurs. After the second test (with the segment made of S690), it will be possible to understand the influence of various steels on the behavior, which should question the rational use of the high-strength steels in this type of bridge applications.

References

[1] Reis A, Pedro J O, Graça A B, Hendy C, Ramoli P, Simões da Silva L and Martins J P 2016 Report on the characterization of relevant parameters of curved plated bridge structures and identification of bridge cases where they can be found RFCS Research Project OUTBURST (Deliverable 2.1) pp 2015-709782

[2] Martins J P, Ljubinkovic F, Simões da Silva L and Gervásio H 2018 Behaviour of thin-walled curved steel plates under generalized in-plane stresses: A review J. of Constr. Steel Research (Elsevier) 140 191-207

[3] European Committee for Standardization (CEN) 2006 Design of steel structures; Plated structural elements Eurocode 3, Part 1-5 (EN 1993-1-5)

[4] European Committee for Standardization (CEN) 2007 Design of steel structures; Strength and stability of shell structures Eurocode 3, Part 1-6 (EN 1993-1-6)

[5] Riks E. 1979 An incremental approach to the solution of snapping and buckling problems Int. J. Solids Struct. 15 524-521