Studia Geotechnica et Mechanica, Vol. XXVIII, No. 2–4, 2006

FULL-SCALE TEST OF CORRUGATED STEEL CULVERT AND FEM ANALYSIS WITH VARIOUS STATIC SYSTEMS

BARTŁOMIEJ KUNECKI

Institute of Civil Engineering, Wrocław University of Technology,
Wybrzeże Wyspiańskiego 27, 50-370 Wrocław, Poland.

Department of Civil and Transport Engineering, The Norwegian University of Science and Technology,
Høgskoleringen 7A, 7491 Trondheim, Norway.

Abstract: Corrugated steel culverts are increasingly being used in road and railway projects as the solutions alternative to small-span concrete and steel bridges. Their construction period is short, and the structures have both technical and economical advantages. This paper describes full-scale static test of an instrumented corrugated steel culvert with 2.99 m span and 2.40 m height. The corrugation is 159 × 50 mm, and the steel thickness is 3.75 mm. The standardized railway loads configuration UIC 71 for Europe was applied at 0.80 m soil cover. The present test was carried out in the Bridge and Road Research Institute in Żmigród, Poland, in 1998. Several full-scale field tests have been performed to validate the long-term performance and great load bearing capacity of these structures, but not very many structures have been tested in controlled conditions in a test facility, such as that described here. The test results were compared with the results obtained from numerical analysis. Two Finite Element Method (FEM) models of tested structure were built within the Cosmos/M software. Various static systems with different boundary conditions were used for each FEM model.

Streszczenie: Przepusty z blachy falistej są coraz częściej wykorzystywane w budownictwie drogowym i kolejowym jako rozwiązanie alternatywne dla małych mostów betonowych i stalowych oraz przepustów betonowych. Krótki czas wznoszenia powoduje, że konstrukcje tego typu charakteryzują się zarówno technicznymi, jak i ekonomicznymi zaletami. W artykule opisano statyczny test pełnowymiarowego przepustu o rozpiętości 2,99 m i wysokości 2,40 m. Blacha o wymiarach fałdy 159 × 50 mm miała grubość 3,75 mm. Zastosowano kolejowe obciążenie normowe według normy europejskiej UIC 71. Opisane badania przeprowadzono w Instytucie Badawczym Dróg i Mostów w Żmigrodzie w 1998 przy naziomie wynoszącym 0,8 m. Aby potwierdzić długą żywotność i wytrzymałość konstrukcji podatnych z blachy falistej, do tej pory wykonano kilkanaście testów w pełnej skali w warunkach polowych, ale niewiele konstrukcji było testowanych w tak kontrolowanych warunkach jak opisana poniżej. Otrzymane wyniki empiryczne porównano z wynikami otrzymanymi Metodą Elementów Skończonych (MES). Wykonano dwa modele numeryczne MES testowanej konstrukcji w systemie Cosmos/M. Dla każdego modelu MES zastosowano inny schemat statyczny różniący się warunkami brzegowymi.

Резюме: Водопропускные трубы из волнистой листовой стали все чаще используются в дорожном и железнодорожном строительстве в качестве альтернативного решения для малых бетонных и стальных мостов, а также бетонных водопропускных труб. Краткое время построения вызывает то, что такие конструкции характеризуются как техническими, так и эконмическими преимуществами. В настоящей статье описан статический тест полноразмерной водопропускной трубы пролета 2,99 м и высоты 2,40 м. Жесть размера заката 159 × 50 мм характеризовалась толщиной 3,75 мм. Была применена железнодорожная нормовая нагрузка согласно европейским нормам UIC 71. Описанные исследования были проведены в Исследовательском Институте Дорог и Мостов в Жмигроде в 1998 году при покровном слое толщины 0,8 м. Для подтверждения долговременности и стойкости податливых конструкций из волнистой стали были выполнены многочисленные полномасштабные тесты в полевых условиях, но лишь некоторые подвергались тестам, какие описаны ниже. Полученные эмпирические результаты были сравнены с результатами, полученными методом конечных элементов (МКЭ). Были выполнены два численных моделя МКЭ тестируемой конструкции в системе Cosmos/M. Для каждой модели МКЭ была применена отдельная статическая схема, отличающаяся предельными условиями.

1. INTRODUCTION

Corrugated steel culverts are being increasingly used in road and railway projects as the solution alternative to bridge and tunnels. Several full-scale fields tests have been carried out to validate the long-term performance and load bearing capacity of these structures. In contrast to the above, only few tests of life-size structures have been performed under fully controlled laboratory conditions.

The experimental data obtained under such conditions allow us to verify software tools used for numerical analysis as well as to optimize and design more economic structures. It is of the key importance to have high-quality test results when designing flexible, long-span buried structures with minimum cover for live railway and road loads.

2. DESCRIPTION OF THE STRUCTURE TESTED

The structure tested was located in the test stand shown in figure 1. The test stand has the form of an 80.0 m long and 12.0 m wide reinforced concrete foundation with a system of anchors and a steel frame. The steel frame serves as a support structure for the system of two hydraulic actuators with a modern control and feeding system ensuring full control over the static and dynamic loads in real time [1].

The culvert tested had the span of 2.99 m, the height of 2.40 m and the length of 14.4 m. Soil cover of 0.80 m was used while testing railway loads. The steel material had the quality FE 360 B FN according to European Standard EN 10025. The minimum yield stress was 245 MPa. The corrugation was 150 × 50 mm, and the steel thickness, 3.75 mm. The steel plates were joined by bolts, 20 mm in diameter, with minimum tensile strength of 830 MPa. The bolts and the joints are shown in figure 2 and the properties of steel plate are listed in table 1 [1].

Table 1

Properties of the steel plate

Name	Plate thickness	Area *A*	Moment of inertia *I*	Section modulus *W*	Radius of gyration *i*
Units	[mm]	[mm^2/mm]	[mm^4/mm]	[mm^3/mm]	[mm]

Value	3.75	4.72	1479.8	55.1	17.7

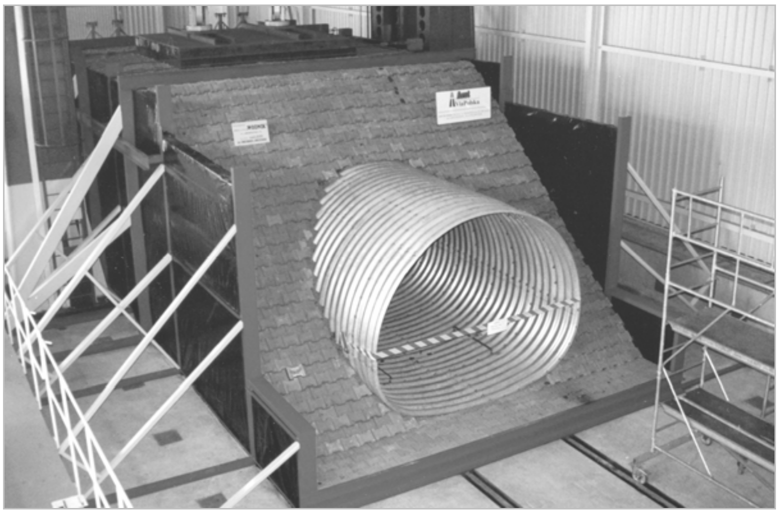

Fig. 1. View of the culvert tested

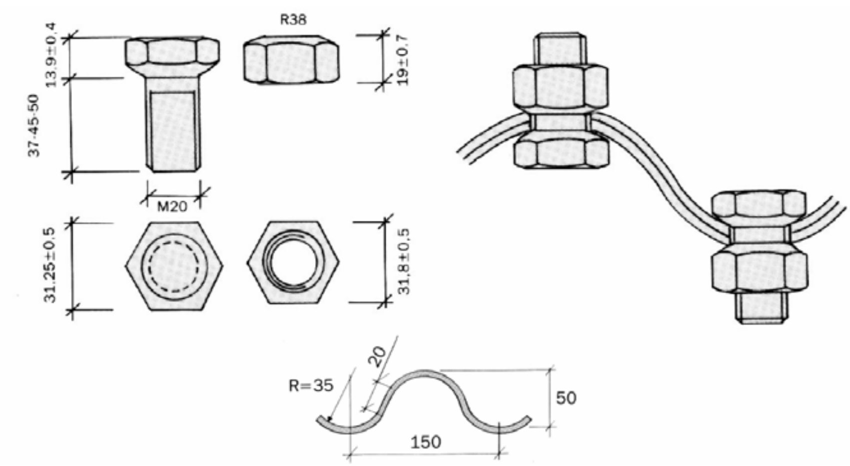

Fig. 2. Cross section of the steel plate and the bolts [mm]

The test bin was 12.0 m long, 5.0 m wide and 4.0 m high. It was constructed of railway sleepers and steel beams. The backfilling material used in the test was a semi-gravel with a maximum grain size of 32 mm. The backfill was placed in the layers of the maximum thickness of 20 cm before compaction. The required degree of compaction was 97% Standard Proctor, except for the 500 mm closest to the structure where

94% Standard Proctor was sufficient. The cross section of the test stand is shown in figure 3 [4].

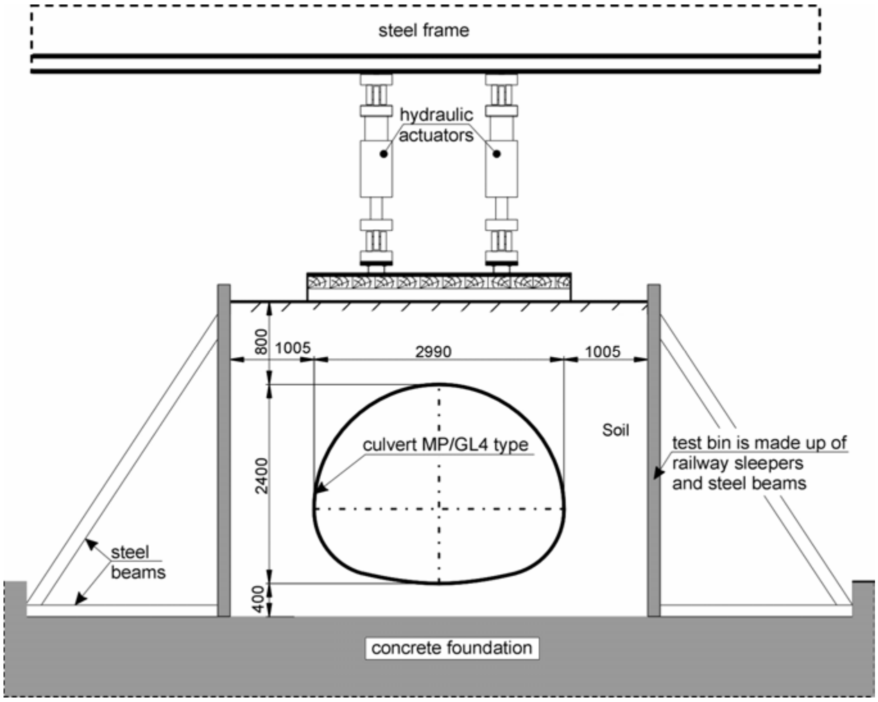

Fig. 3. Cross section of the culvert tested [mm]

3. LOADS

The loads from two hydraulic actuators were distributed through two layers of wooden sleepers and a 20 mm thick steel plate of the area of 2.60 m × 3.15 m. The cross section and the load distribution system are shown in figure 4 [2].

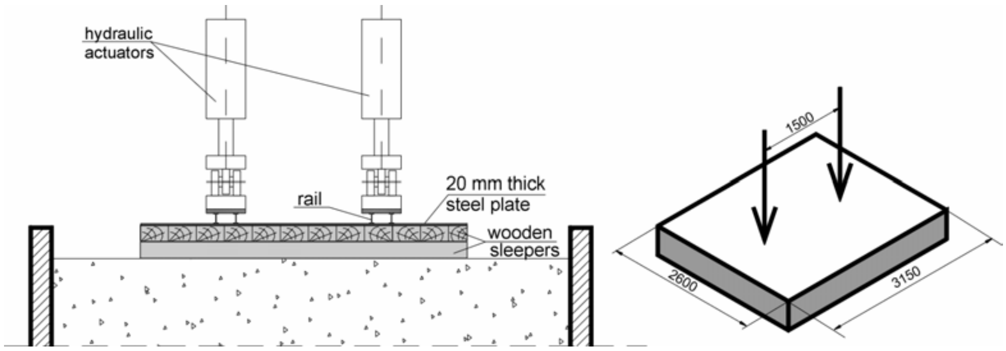

Fig. 4. Cross section and the load distribution system [mm]

The standardized configuration of railway loads UIC 71 for Europe was applied. Due to the distribution effect of rails, sleepers and ballast bed, the axle loads from the locomotive (4 × 250 kN) produce a uniform area load which equals approximately 52.0 kN/m^2 at a base of the ballast bed at the depth of 0.5 m [1]. At the depth of 0.8 m the uniform load is 51.0 kN/m^2. The dynamic load factor is 1.37 at 0.8 m cover, according to the formula given by the European Standard. The resulting pressure which was applied during railway standard static test equalled 51.0 × 1.37 = 69.87 kN/m^2 [1].

A real force used in each actuator is listed in table 2. Three standard static loads were applied at regular time intervals (about 20 minutes) [1].

Table 2

Loads

Name	Number of loads	Soil cover *h*	Dynamic factor ϕ	Area of rigid plate	Total force *F* for two actuators	Total pressure *p*
Units	[–]	[m]	[–]	[m^2]	[kN]	[kN/m^2]
Standardized loads	3	0.8	1.37	8.19	572	69.84

4. INSTRUMENTATION AND MEASUREMENT

The instrumentation comprised the following elements:

Strain gauges. Strain gauges were placed at 14 locations inside the steel structure. Two gauges were fitted to each location, one at the top of the corrugation and one at the bottom (a total of 28 strain gauges). The location of all strain gauges is shown in figure 5a. This configuration allowed axial and bending strains to be measured. Dummy gauges were installed to provide temperature compensation. Electroresistant strain gauges of Hottinger Baldwin Masstechnik (HBM) 6/120LY41 type were installed. Strain gauges had 6 mm measurement base, 120 Ω resistance and the factor *k* of 2.02. The measurements were preformed with the tension-metric bridge UPM 100 also from HBM. The UPM 100 was connected to Macintosh computer equipped with "Beam" software [1].

Earth pressure cells. Earth pressure exerted onto the steel structure by the surrounding soil was measured by the earth pressure cells. Eight of them were installed at the steel structures, about 6 cm from steel plates. Additionally two earth pressure cells were installed at the top of both sides of the structure at the distance of 1.5 m from the axis of symmetry. The location of the all earth pressure cells is shown in figure 5b. Each cell was protected by 3 cm layer of dry sand and durable foil. The earth pressure cells during installation are shown in figure 6. Magneto-elastic pressure cells from Wrocław University of Technology were used. Recording system PPN-3 with "Dynusing" software was used to collect data from earth pressure cells [1].

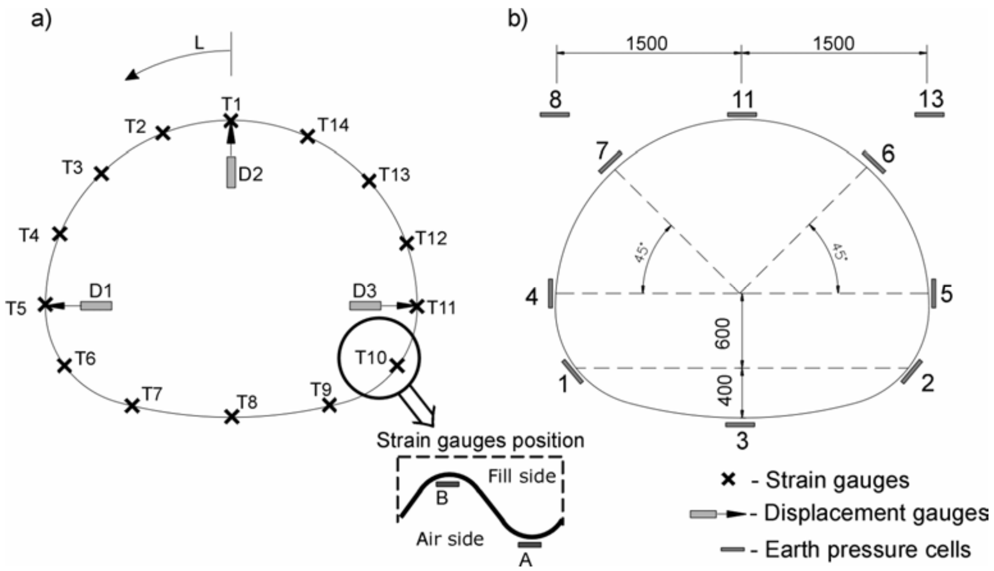

Fig. 5. Instrumentation of the culvert tested:
a) the strain and the displacement gauges, b) the earth pressure cells [mm]

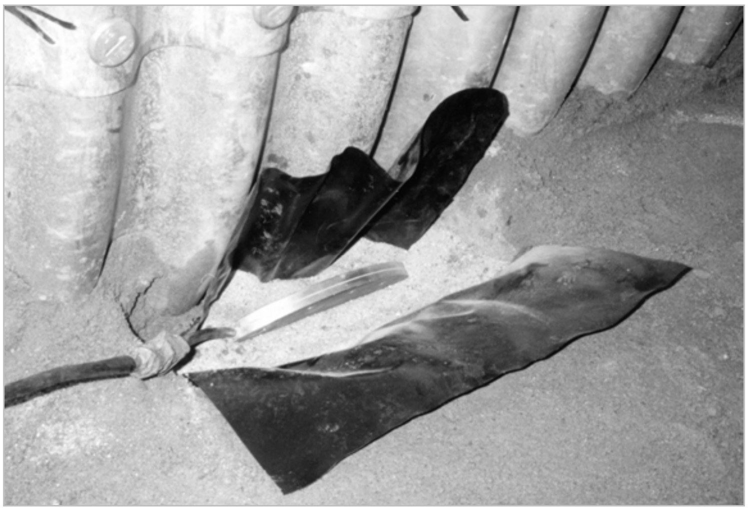

Fig. 6. Earth pressure cell during installation

Displacement gauges. Three displacement gauges were installed in the same plane (one vertical gauge and two horizontal gauges). The displacement gauges allowed a relative displacement of the metal culvert to be measured. The maximum vertical and horizontal diameters were measured by displacement gauges. The W50TS type inductive gauges were used from HBM. Each gauge had a measuring range of ±50 mm and the precision class of 0.4. The locations of all displacement gauges are shown in figure 5a [1].

After each load strain gauges, earth pressure cells and displacement gauges were resetted.

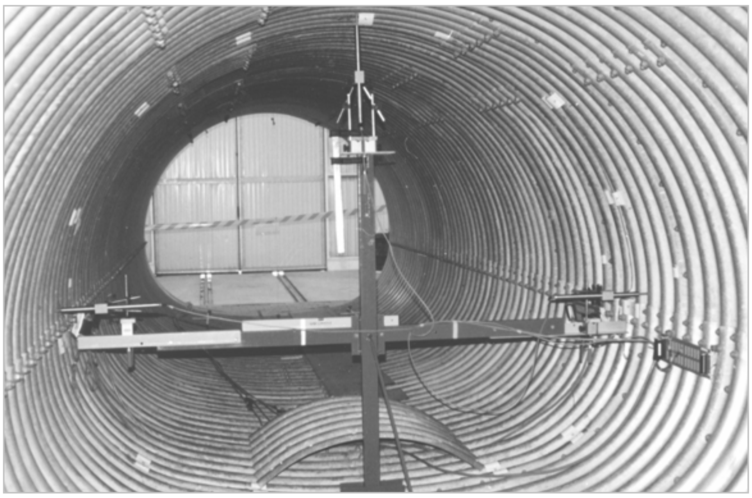

Fig. 7. Displacement gauges

5. TEST RESULTS

All the results of displacement and earth pressure tests are listed in tables 3 and 4, and the recordings of strain gauges are shown in figure 13. The average values from three loads are given. All of them were obtained 20 s after reaching a full load assumed.

The stresses in steel structure were calculated based on strains taking into consideration that the elastic modulus of steel is E_s = 205 GPa. The stresses at each measuring point (paired strain gauges per a point) were allocated to axial stresses and bending stresses according to the following equations:

$$\sigma_a = \frac{\sigma_A + \sigma_B}{2}, \tag{1}$$

$$\sigma_b = \frac{\sigma_A - \sigma_B}{2}, \tag{2}$$

where:

σ_a – the axial stresses,

σ_b – the bending stresses,

σ_A – the stresses at the point *A* (at the crest of corrugation),

σ_B – the stresses at the point *B* (at the valley of corrugation).

6. MODELLING OF THE STRUCTURE

The analyses of the soil–structure system were carried out using the computer software Cosmos/M system. This software allows the simulation of live loads together with deadweight of soil and structure. The results presented show the calculation which included the deadweight of soil and structure.

The culvert was modelled by the 4-node shell elements, and the soil – by 8-node solid three-dimensional elements. The whole FEM model includes 2 560 SHELL4 elements for modelling the steel structures, 20 480 SOLID elements for modelling the soil and 1 280 elements for modelling the wooden bin. Static models of soil–structure systems are shown in figure 8 and the finite elements for model 2 are presented in figure 9.

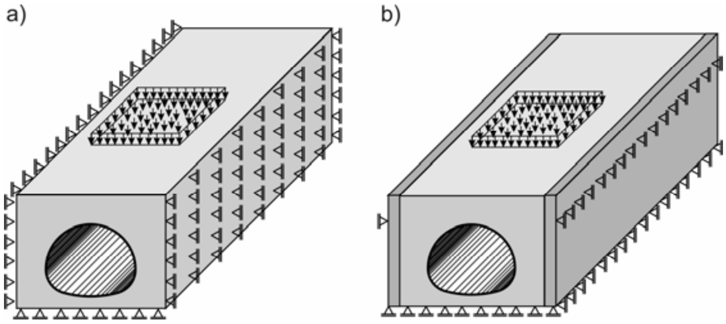

Fig. 8. Static systems of FEM models:
a) system 1 with rigid test bin, b) system 2 with wooden test bin

The modulus of elasticity for the soil (semigravel) was assumed to be $E = 90.0$ MPa, the angle of internal friction: $\Phi = 35°$ and the cohesive strength $c = 0.1$ Pa. The Poisson ratio for soil was $\nu = 0.2$ and the density $\gamma = 20.0$ kN/m^3. The modulus of elasticity for steel plates was $E = 205$ GPa, the Poisson ratio $\nu = 0.3$ and the density $\gamma = 78.5$ kN/m^3. The parameters for soil were determined from in situ test, while the parameters for steel being assumed according to Polish Standards.

For the sake of convenience, the thickness of steel plate was assumed to be equal to the height of corrugation, i.e., 50 mm. The modulus of elasticity was accordingly reduced to keep the required value of *EI*. The reduced modulus of elasticity for steel plate was calculated from the following equation [3]:

$$E_r = E \cdot \left(\frac{t}{50} \right) = 15.375 \text{ GPa} , \tag{3}$$

where:

$t = 3.75$ mm – the thickness of steel plate,

E – the modulus of elasticity for steel.

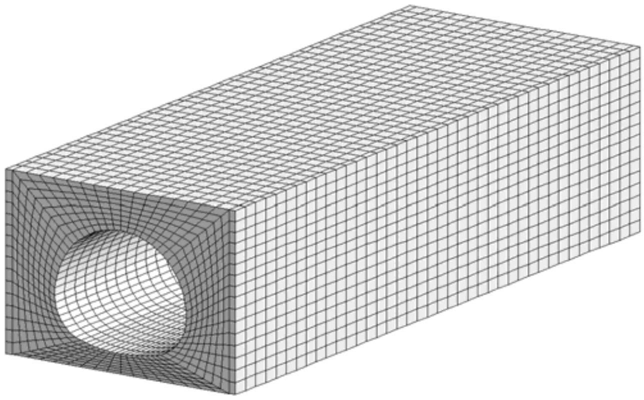

Fig. 9. FEM model for system 2

Element SHELL4. SHELL4 is a 4-node quadrilateral thin shell element with membrane and bending capabilities that make it ready for the analysis of three-dimensional structural and thermal models. The shear deformation effect is neglected. Six degrees of freedom per node (three translations and three rotations) are considered for structural analysis. The element of constant thickness is assumed to be isotropic for structural problems. Stress components including von Mises stresses are available in the element coordinate systems at the centroid of the element for top and bottom fibres [5].

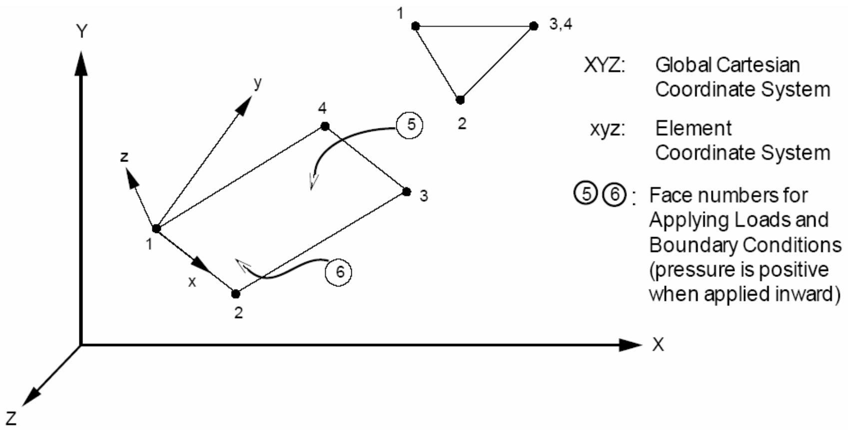

Fig. 10. The SHELL4 element

Material properties used in SHELL4 element are as follows: EX – the modulus of elasticity in the first material direction, NUXY – Poisson's ratio relating the first and the second material directions, DENS – the density. A linear elastic material model was used to model steel. The nodal input pattern for this element is shown in figure 10 [5].

Element SOLID. SOLID is an 8–20-node three-dimensional element for the analysis of structural problems. Three translational degrees of freedom per node are considered for structural analysis. The nodal input pattern is shown in figure 11 for the local node numbering of a 20-node element. Nine constants have to be defined in the case of using orthotropic or anisotropic model. The nine values are used to determine the coordinates of three points defining the material coordinate system. The first direction of the material coordinate system (denoted by *a* in figure 11) is defined by a vector connecting the point 1′ with the point 2′. The *b*-axis (the second material direction) lies in the plane of the three defined points and goes from the *a*-axis toward the third point. The *c*-axis (the third material direction) completes a right-hand Cartesian coordinate system. Stress components in either the global or element coordinate direction including the von Mises stresses are available at all nodes and the center of the element. The directions of the stress components are shown in figure 12 [5].

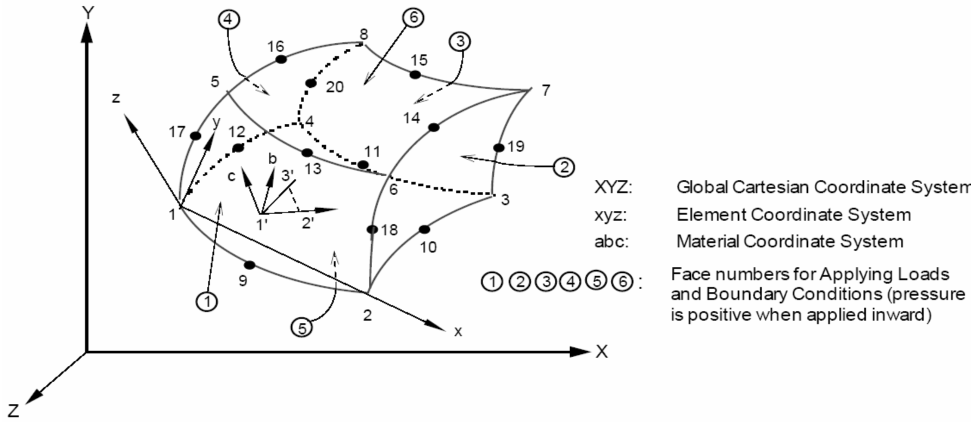

Fig. 11. The SOLID element

Material properties used in SOLID element are the following: EX – the modulus of elasticity in the first material direction, NUXY – Poisson's ratio relating the first and second material directions, DENS – the density, FRCANG – the angle of internal friction, COHESN – a cohesive strength, SIGYLD – the yield stress for bilinear elastoplastic models, ETAN – the tangent modulus for bilinear elastoplastic models [5]. The Drucker–Prager elastic–perfectly plastic model was used for modelling the semi-gravel.

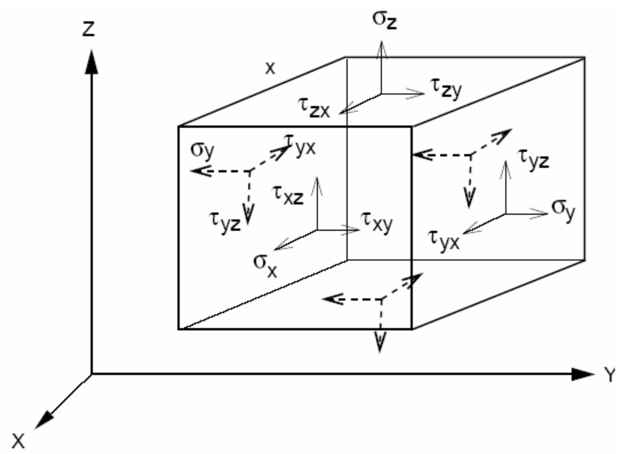

Fig. 12. The directions of the stress components in the SOLID element

7. THE RESULTS OF FEM ANALYSIS

The measured and computed bending moments in the steel structure are shown in figure 13a. For the system 2 their distribution is seen to be in close a agreement with the distribution of test results. The values numerical bending moments for the system 2 are higher than these for the system with the rigid test bin.

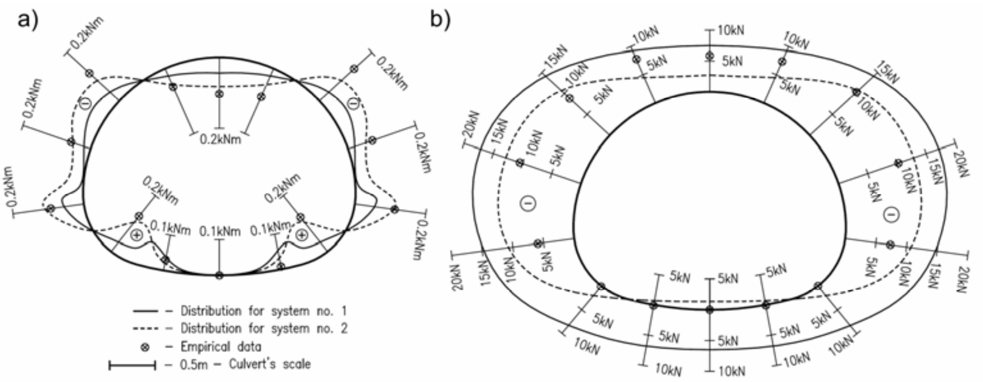

Fig. 13. Distribution of: a) the bending moments [kNm], b) the axial forces [kN]

The measured and computed axial forces in the steel structure are shown in figure 13b. The predicted axial forces are not in good agreement with the measured values, being higher at the crown and on the sides of the culvert and significantly higher at the bottom of the structure in the case of system 1. The distribution of axial forces in the system 2 is in closer agreement than the distribution in the system 1, being slightly lower at the crown, slightly higher on the sides and slightly higher at the bottom of the culvert in comparison with the test results.

The measured and numerical displacements in the steel structure are listed in table 3. The displacements in selected points are seen to be in close agreement, especially in the case of FEM model 2. The difference between test and FEM analysis (model 2) is 0.17 mm (7.3% of the value tested) in the case of vertical displacement and 0.24 mm (12.1% of the value tested) in the case of horizontal displacements.

In the second model, the predicted and measured earth pressures at selected points are generally in reasonable agreement. The predicted earth pressures at the point 11 (over the crown) are not in good agreement with the measured values, being significantly lower (table 4) for both models. This indicates that the steel shell behaves like flexible structure and the high value obtained in the test at the point presented probably results from error (the gauge presented was located directly under hydraulic actuators).

Table 3

Comparison of the displacements at selected points

Point	*D*1	*D*2	*D*3
Units	mm	mm	mm
Average test results	0.95	–2.33	1.03
FEM for model 1	0.16	–1.26	0.16
FEM for model 2	0.87	–2.16	0.87

Table 4

Comparison of the earth pressures

Point	1	2	3	4	5	6	7	8	11	13
Units	kPa	kPa	kPa	kPa	kPa	kPa	kPa	kPa	kPa	kPa
Average test results	52.7	51.6	0.1	32.1	27.1	21.4	26.6	31.6	26.7	50.3
FEM for model 1	29.3	29.3	4.7	22.0	22.0	16.5	16.5	15.7	3.2	15.7
FEM for model 2	37.0	37.0	0.0	25.7	25.7	20.8	20.8	18.0	9.1	18.0

Cosmos/M system allows several results to be presented graphically. The following distributions for two FEM static systems are shown in figures 14–17:

– the von Mises stress σ_{von} for steel,

– the normal stresses in the *x*-direction (σ_x) in the soil surrounding buried structure,

– the normal stresses in the *y*-direction (σ_y) in the soil surrounding buried structure,

– the displacements with deformation.

Figures 15 and 16 show the distribution in the middle cross-section of the system (under live load). For each figure the same scale was used. The von Mises stress σ_{von} is calculated from the stress components as shown below [5]:

$$\sigma_{\text{von}} = \sqrt{\tfrac{1}{2}[(\sigma_x - \sigma_y)^2 + (\sigma_x - \sigma_z)^2 + (\sigma_y - \sigma_z)^2 + 3(\tau_{xy}^2 + \tau_{xz}^2 + \tau_{yz}^2)]}\,, \tag{4}$$

where:

σ_i – the normal stresses (i – x, y, z direction),
τ_{xy} – the shear stress in the x–y plane,
τ_{xz} – the shear stress in the x–z plane,
τ_{yz} – the shear stress in the y–z plane.

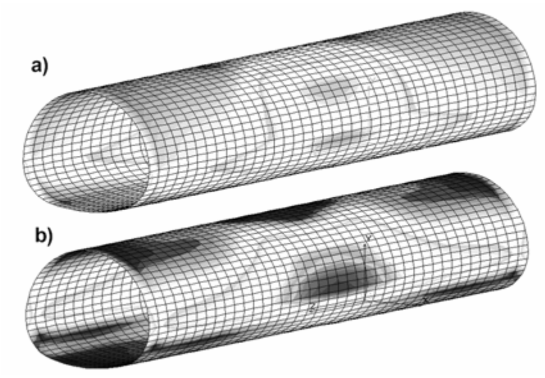

Fig. 14. Comparison of the von Mises stresses in steel for:
a) FEM system 1, b) FEM system 2

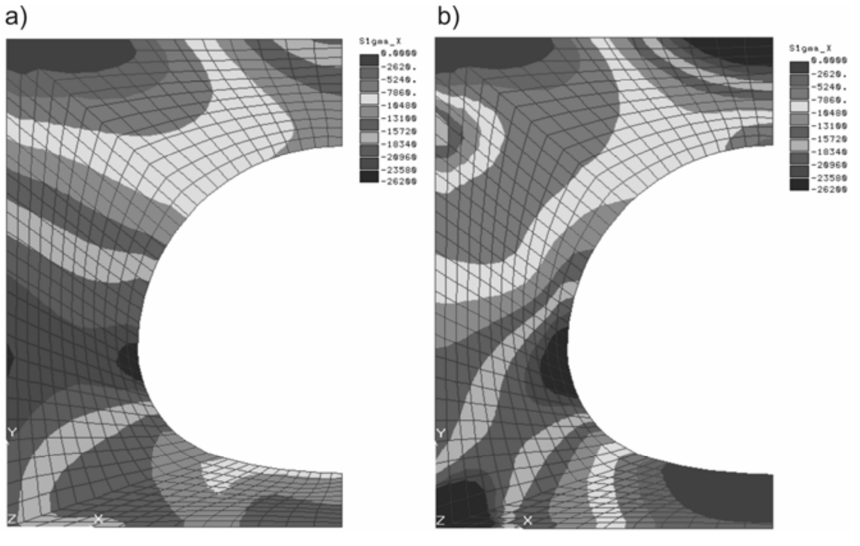

Fig. 15. Normal stresses in surrounding soil in the x-direction obtained for:
a) FEM system 1, b) FEM system 2 [N]

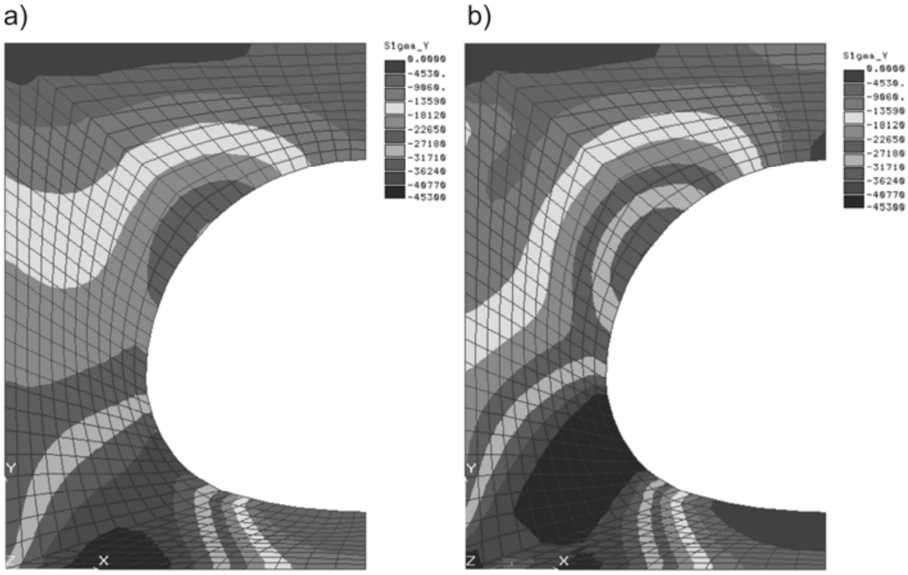

Fig. 16. Normal stresses in surrounding soil in the *y*-direction obtained for:
a) FEM system 1, b) FEM system 2 [N]

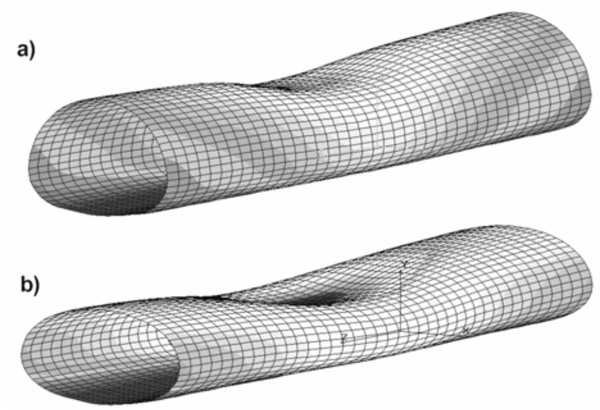

Fig. 17. Comparison of the displacement with deformation for:
a) FEM system 1, b) FEM system 2

8. SUMMARY AND CONCLUSION

The comparative analyses presented prove that the assumed static system of the structure tested has fundamental influence on the results obtained in the numerical analyses. Simplification of static system, i.e., assumption that the cover bin on the test stand is stiff, has caused distortion of numerical results in comparison with results obtained from tests. Comparison of the results of the displacement at three crucial points of the structure (table 3 and figure 17), stresses in the soil (figures 15 and 16) and internal forces (figures 13 and 14) indicates that distortion was both qualitative and quantitative in character.

Non-uniform vertical pressure under the plate can be the cause of the differences between measured and calculated values. Two actuators on a 20 mm steel plate will not produce a uniform (standard) vertical pressure under the plate. Measurements of a vertical deformation at different positions across the loading plate (under edges, corners and centre, etc.) would have shown that the deformations are not uniform.

In this kind of structures, a crucial issue is the interaction between structure and surrounding soil. Close proximity of non-ideal rigid walls, which were made from the wooden railway sleepers, had significant impact on the behaviour of the structure tested which should be taken into account when modelling the static system.

The full-scale test, analyses and conclusions presented will come in useful during preparation for next future full-scale laboratory tests. We should try to eliminate the influence of the test stand if it is possible, otherwise we should make detailed numerical models allowing accurate reflection of real environmental conditions.

REFERENCES

[1] KUNECKI B., *Full-scale laboratory static tests of steel corrugated pipe-arch culvert*, Proceedings of the 8th International Conference: *Shell Structures, Theory and Application*, Gdańsk, October 2005, 589–593.

[2] KUNECKI B., KUBICA E., *Full-scale laboratory tests and FEM analysis of corrugated steel culverts under standardize railway load*, Archives of Civil and Mechanical Engineering, 2004, Vol. 4, No. 4, 41–53.

[3] VASLESTAD J., MADAJ A., JANUSZ L., BEDNAREK B., *Field measurements of an old brick culvert slip lined with a corrugated steel culvert*, Transportation Research Record: Journal of the Transportation Research Board, 2004, No. 1892, TRB National Research Council, Washington D.C., 227–234.

[4] KORUSIEWICZ L., KUNECKI B., *Experimental and numerical analysis of internal forces in steel culverts of the multiplate type*, Proceedings of the 7th International Conference: *Shell Structures, Theory and Application*, Gdańsk, October 2002, 137–138.

[5] RUSIŃSKI E., *Finite Elements Method – Cosmos/M system* (in Polish), Publication of Transport and Communication, Warsaw, 1994.