Budapest University of Technology and Economics
Faculty of civil engineering
Department of Structural Mechanics

Numerical and experimental based aerodynamic stability study of bridge decks

Theses of the dissertation

Written by:
Gergely Szabó
Pont-Terv Ltd.
Department of Structural Mechanics

Scientific supervisor:
József Györgyi
Department of Structural Mechanics

CFD expert:
Gergely Kristóf
Department of Fluid Mechanics

Budapest, 2013

1. Background of the study, objectives

In this dissertation aeroelastic stability of bridge decks is investigated by means of wind tunnel tests and numerical simulation. The numerical study of the wind related problems is aided by the development of the CFD (Computational Fluid Dynamics) software and the significant increase of computer power. Aeroelastic study of bridges is of primary relevance of slender structures or structural parts, which are vulnerable to wind loading. In case of long span bridges, flutter is the most dangerous type of instability; therefore flutter assessment becomes a primary design task. The most widespread approach for bridge deck flutter study is still the wind tunnel test. Basically section models are used that represent a section of the whole bridge deck. In special cases (i.e. large span, complex dynamic mode shapes, strong three-dimensionality of the wind loading) full aeroelastic wind tunnel models are made, which are extremely expensive. In addition to that, all the fluid and structural related similarity laws can not be entirely kept at the same time.

By using CFD, flutter can be solved numerically as well. In literature, only 2D simulations are found. This approach can be regarded as a virtual counterpart of the section model tests. According to literature, the reliability of the 2D approach should be further improved by means of a great number of CFD simulations as well as section wind tunnel tests until it can be considered as a tool in bridge design.

Having investigated the methods and results available in literature, two main approaches seemed interesting and important to deal with. The first main field is the 2D CFD related research work, in which the methods found in literature are to be applied and further developed. The main goal is to work out a 2D numerical approach that provides quick results for practice. The second main field is the development of a three-dimensional numerical approach that can be regarded as the virtual counterpart of the full aeroelastic wind tunnel tests. Such simulations can not be found for bridges in literature, therefore the necessary pieces of information were collected from other fields. Seemingly the aero-industry and other fields (i.e. bio-mechanics) exploit such advanced simulations. In this part, the main goal is the development of a three-dimensional numerical approach for aerodynamic stability study of bridge decks.

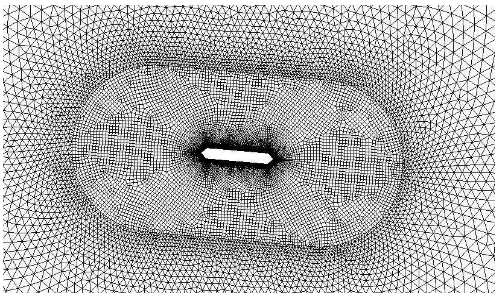
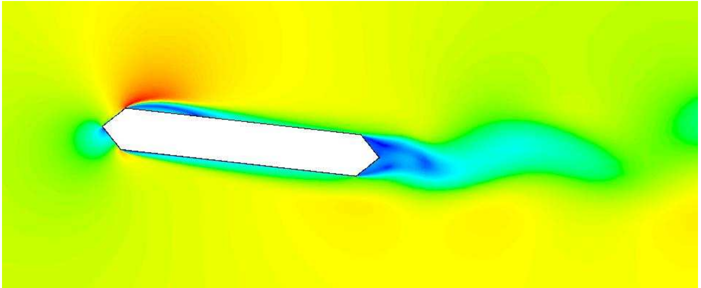
The main objectives of the dissertation are summed up below:

- For aeroelastic study of structures, both structural analysis and CFD simulations are necessary. Before dealing with flutter itself, fixed bodies were considered by using CFD. The accuracy and reliability of CFD for civil engineering problems seemed to be important to judge. In order to test the CFD, fundamental studies were to be performed. A circular and a rectangular shaped body was scheduled to be simulated for this purpose.
- The experiences gained by investigating fixed bodies can be exploited to consider moving bodies. We planned to get acquainted with the coupled methods (FSI, *Fluid Structure Interaction*) as a solution to flutter. To do so, the appropriate technique needed to be selected based on literature and testes on a simple case.

- Based on the tested coupled method, idealized and realistic bridge deck sections were to be investigated. The direct and indirect methods were both planned to be applied. A section wind tunnel test program was scheduled to carry out as an extended validation of the numerical approaches. Special care should be taken of the dynamic CFD mesh. Although ready-to-use dynamic mesh capabilities are offered by the commercial software, a main goal was to work out an effective dynamic mesh method, with which each mesh node motion is controlled, and the cell distortion can thus be minimized properly.
- After dealing with the 2D CFD models and the section wind tunnel tests, the main purpose was the development and the application of a three-dimensional approach. All the critical issues and aspects of this method was planned to study and reveal. The target software was the Ansys-CFX and the Ansys mechanical classic, which can be coupled by using the built-in module (MFX, *Multifield Solver*) of the software. The user can only utilize a black box this way, therefore an alternative coupled method was also planned to work out. The Fluent was utilized for this purpose with UDF (*User Defined Functions*).
- Besides the investigation of the coupled methods, a new approach was also planned to develop, by which the advantages of the theory of the flutter derivatives and that of the three-dimensional CFD simulations can be combined. A full aeroelastic wind tunnel model was planned to carry out in order to validate both the direct and indirect techniques.
- The developed approaches were to be tested on bridge structures already built on Hungary.

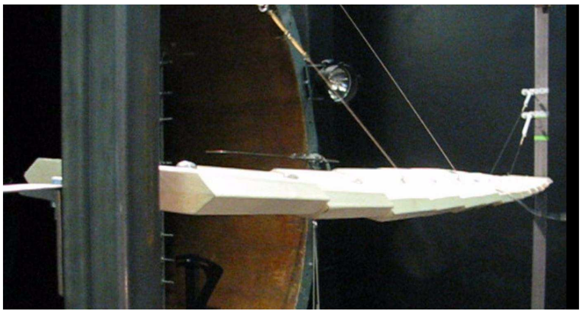
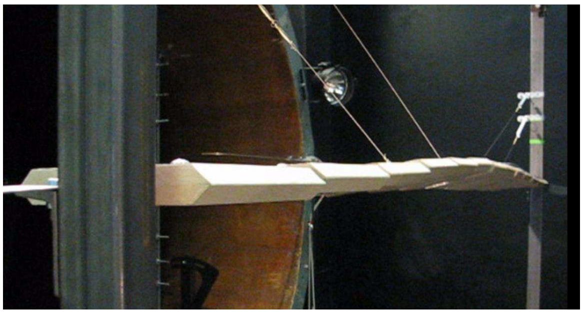
2. Details of the research program

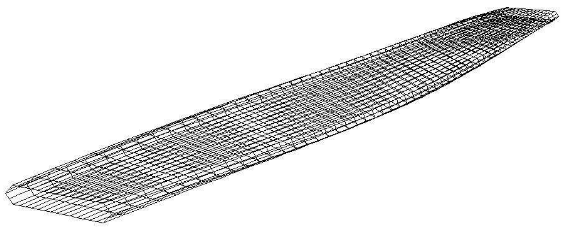
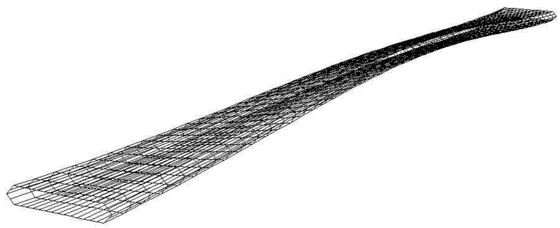
Considering the objectives, the research program was scheduled. In order to get familiar with the CFD, circular and rectangular fixed bodies were planned to be analyzed. In case of the circle, the Reynolds-number effect seemed to be important to investigate by considering a range of wind velocity. In case of the rectangle, however, the Reynolds-number effect is dominated by the side ratio effect. Comparisons had to be done with the Eurocode only, no wind tunnel tests were designed for this case. A key issue was the investigation of the role of the turbulence models and the numerical grid.
The next step had to be the application of CFD for flutter by utilizing the Fluent software. At first the case of the 2DOF flat plate immersed in airflow had to be considered, thus comparison with analytical values could be done. In order to test the numerical approach thoroughly, the direct and the indirect methods were to be used. The direct method required the development and application of coupled simulations. The coupled method had to be tested in terms of time step size sensibility before using it for practical problems. After validating the coupled (direct) and the forced oscillation (indirect) methods, bridge cross sections can be considered. In case of numerical CFD simulations of flutter, the mesh grid points have to move, leading to certain cell distortion that is unpleasant in terms of convergence. A special new meshing approach had to be worked out, by which the good mesh quality can be preserved throughout the whole computational time. The 2D CFD simulations were to be carried out for different cross sections; two generalized and two realistic bridge shapes were to be considered.
Besides flutter assessment, the Strouhal-number had to be also evaluated. A test program had to be performed in order to compare the measured and the calculated critical wind speeds. In the picture below the distorted numerical grid is shown around the streamlined cross section (on the left). On the right, the velocity contour plot is shown for the same case.

Based on the experiences gained on the 2D simulations, the 3D simulations seemed to be possible to perform. At first, the Ansys software had to be utilized, because the coupling of the fluid flow and the structural behaviour is automatically handled. At second, the Fluent software combined with C programming had to be used, by which the computational time had to be reduced and the black box of the Ansys had to be checked. In this case the commercial software was planned to be used to get the aerodynamic forces, but the structural motion had to be handled by an own written program. As in case of the 2D CFD simulations, the CFD mesh grid point motion had to be controlled with an own program of the 3D CFD.

In the two pictures below the flutter motion of the full aeroelastic wind tunnel model can be seen at 10 m/s wind velocity. The further two pictures below show the same case by using the 3D coupled simulation performed with the Fluent software.

As an alternative approach, the 3D simulations and the forced oscillation method seemed to be reasonable to combine into a new technique. In this novel approach, the 3D bridge deck needed to be given 3D forced oscillation according to the relevant mode shapes. The calculated aerodynamic forces were to be processed properly in order to introduce the new terms of the modal flutter derivatives. The three-dimensional numerical methods had to be validated based on the results of the full aeroelastic wind tunnel tests, in order to ascertain about their reliability. The developed and validated methods were to be applied to real bridge structures already built in Hungary, proving their applicability in industrial projects.

3. Results and theses of the dissertation

The theses of the dissertation are concluded as follows. After each thesis I briefly refer to the belonging research activity that can be found in the proper chapters of the dissertation.

1. Thesis

Compared to the approaches found in literature an extended study was carried out, in which a 2DOF flat plate immersed in fluid flow was investigated by means of both direct (coupled fluid-structure interaction) and indirect (flutter derivatives, forced oscillation) methods. The calculated critical wind velocities could be compared more detailed this way. It was stated that in case of the flat plate with the given properties (by using the direct method) the time step size had to be around ten times lower than in case of external dynamic loading in order to maintain numerical stability. This means that the proper time step size in the investigated case was the cycle time belonging to the relevant (rotational) mode shape divided by 200. The flutter derivatives of the flat plate were extracted by means of forced oscillation method. It was stated that the flutter derivatives are less sensitive to the choice of the turbulence models than the parameters of the fixed bodies. The critical wind speed values calculated by using the direct and the indirect methods and determined by means of analytical theory were in excellent agreement.

This thesis consists of the following tasks:

The explicit coupling technique was tested in terms of time step size by using quasi-static aerodynamic forces that are only functions of the angle of attack. A program was written in order to further test the explicit method. This program was integrated into the CFD software, thus the numerical based aerodynamic forces were applied in this case. Both the direct and indirect numerical methods as well at the analytical theory showed good coincidence, proving the applicability of the explicit method.

Belonging chapter: 4.
Belonging publication: [4]

2. Thesis

Compared to the approaches in literature that requires much computational time, a numerical method was worked out for the civil engineering practice, which can be used for aerodynamic stability assessment of bridge decks. In order to develop the method, a wind tunnel test program was designed and carried out (in cooperation with the Department of Fluid Mechanics). The test program involves a number of different bridge deck sections. The aerodynamic behaviour of these cross section shapes was measured. The test sections were simulated by CFD. An effective three-cell zone CFD meshing strategy was worked out, with which the cell distortion can be limited in case of dynamic meshes. Based on the CFD results validated with the wind tunnel test results, suggestions were given as to the numerical specifics (turbulence models, time step size and numerical grid). Based on the calculated critical wind speed values, it was stated that in case of an aerodynamically stable cross section the presence of a vehicle alters the shape and means negative influence on the aerodynamic stability.

This thesis consists of the following tasks:

Numerical flow simulation was carried out for circular and rectangular fixed bodies in order to test the CFD software. Detailed comparison was made with the Eurocode. In the extended study (in case of the circle a Reynolds-number range, in case of the rectangle a side ratio range was considered) the SAS turbulence model was utilized at first in literature. Based on the results the SAS model can be proposed for time dependent flow related problems in case of fixed bodies. The SAS model proved to be appropriate to resolve the Karman-vortices, but less sensitive to the numerical grid than the LES model.
The critical wind speeds were determined by means of direct (explicit coupled simulation) and indirect (forced oscillation method) approaches. It was stated that the results of both methods were close to that of the wind tunnel tests; therefore the applicability of the explicit coupling technique was justified. Unlike the fixed bodies, the k-ε turbulence model provided acceptable accuracy (15%) for civil engineering practice in case of moving and moved bodies.
The shape modification phenomenon was investigated by considering a 30 wide and 4 m height bridge deck with a 4 m height and 2,5 m wide vehicle on the top of it. Two positions were considered; the object stands on the right and the left emergency lanes. By investigating the damping of the mode shapes it was revealed that the rotational damping drops significantly. In addition to that, the translational damping also turns to be negative on a certain reduced wind velocity range, which was positive in the no-vehicle case.

Belonging chapter: 5.
Belonging publication: [6, 15]

3. Thesis

A novel method was developed for aeroelastic stability assessment of bridge decks. In this method the fluid-structure interaction was investigated by using three-dimensional coupled simulations. The results of the simulation and that of the aeroelastic wind tunnel tests were in good agreement, therefore the novel method was proposed to check or replace the aeroelastic wind tunnel tests. By using this method, special structures can be studied in a numerical way. The three-dimensional approach can model the fluid-structure interaction more accurately than the two-dimensional techniques.

This thesis consists of the following tasks:

A fully aeroelastic wind tunnel model was designed and tested (in cooperation with the Department of Fluid Mechanics). The critical wind speed was established. The equivalent FEM shell model of the wind tunnel model was constructed. The CFD mesh around the bridge contour was made and optimized in order to reduce the computational time. A 2D fine mesh was also constructed for validation. The flutter derivatives were extracted from the fine 2D mesh and the coarser 3D mesh. The 3D CFD mesh contains boundary layer cells with a size of the half thickness of the considered bridge deck. The flutter derivatives belonging to the 2D and the 3D CFD meshes showed good agreement.

Three-dimensional coupled simulation was performed and the critical wind speed was calculated that was in good agreement with that of the wind tunnel tests (10% difference). The computational time necessary was 8 days for a single wind loading case.
In order to reduce the computational time, a program was written by involving the modal analysis. The critical wind speed was calculated with the same order of accuracy as before, but in a significantly less computational time; a single wind speed case required only 4 hours.

Belonging chapter: 6.
Belonging publication: [14]

4. Thesis

A novel method was developed for aeroelastic flutter calculation in the frequency domain, introducing the modal flutter derivatives. The method is based on the three-dimensional forced oscillation of the bridge deck combined with the modal analysis. By using the proposed novel method, the advantages of the indirect approaches (quasi-analytical) can be kept and the three-dimensionality of the fluid-structure interaction can be taken into account at the same time.

This thesis consists of the following tasks:

The mathematical formulations belonging to the 2DOF flutter were reformulated, leading to the quasi three-dimensional method that can be found in literature (in other forms).
The aerodynamic force matrix of the quasi three-dimensional method was further developed, in order to use the 3D CFD simulations instead of the 2D ones for deriving the elements of the mentioned matrix.
Analogously to the 2D flutter derivatives, new terms were introduced, called modal flutter derivatives.
The 3D CFD model of the aeroelastic wind tunnel model that was already used in the 3rd thesis was given a forced oscillation according to the modal derivatives method. The modal flutter derivatives were extracted and the critical wind speed was established. The critical wind speed was in good agreement with that of the wind tunnel test and the three-dimensional coupled simulation.
The role of the mode shapes on the critical wind speed was investigated by means of analytical method. It was revealed that the relevant rotational and heave mode shapes provides accurate results (difference in 1%), all the other modes had an insignificant impact.

Belonging chapter: 7.
Belonging publication: [3]

5. Thesis

Two bridge structures built in Hungary were investigated in terms of aerodynamics. In case of the Móra Ferenc bridge it was shown, that there was a 80% difference between the results of the three-dimensional and the quasi three-dimensional methods, highlighting the importance of the more precise models. An approximation technique was proposed, with which the influence of the structural motion on the aerodynamic forces can be evaluated. In case of the pedestrian bridge in Szolnok the quasi three-dimensional and the modal flutter derivatives methods were applied for the calculation of the critical wind speed. The results of the two methods were close to each other. It was shown that in case of a pedestrian arch bridge with a bridge deck consisting of three tubular steel beams, two heave mode shapes were important (besides the relevant rotational one) in the aerodynamic behaviour.

This thesis consists of the following tasks:

A three-dimensional CFD model was set up for a bridge structure in a free cantilever construction stage. The aerodynamic forces were calculated in fixed and oscillated cases. A two-dimensional CFD model was also made for the same structure. The aerodynamic forces were calculated in fixed and oscillated cases.
By comparing the results, it was shown that in case of fixed simulation there were significant differences. In case of the oscillated cases, however, the differences remained moderated that was explained with the lock-in effect.
Based on the aerodynamic forces the dynamic vertical displacement of the end of the cantilever was evaluated. It was shown that in fixed case the three-dimensional approach provided less value by 80 percent than the quasi three-dimensional one.
An approximation method was proposed for the calculation of the fluid-structure interaction in case of vortex shedding. In the proposed method the 3D CFD model was given a three-dimensional forced oscillation. The dynamic motion of the end of the cantilever was calculated based on the determined aerodynamic forces.
The 3D CFD model of the pedestrian bridge deck was made and the modal flutter derivatives method was applied. The calculated critical wind speed was close to that of the quasi three-dimensional method, proving the applicability of the modal approach for industrial problems as well.
The quasi three-dimensional method was used to analyze the role of the mode shapes, as was done in case of the aeroelastic wind tunnel model. It was shown that besides the relevant torsional mode, two heave mode shapes were important in the dynamic behaviour.

Belonging chapter: 8.
Belonging publication: [2, 5, 16]

Open questions, further research plans

In this dissertation, aerodynamic studies of bridge structures by using numerical simulation were done. The first main topic was the 2D simulations. The numerical parameters were proposed to choose properly so as to obtain reliable results. It is highly recommended to further develop and thoroughly check 2D approaches, for instance by considering new types of cross sections or involving (atmospheric) turbulent fluctuations.

The three-dimensional method seems very prosperous. By using it, there can be a possibility to replace or check the full aeroelastic wind tunnel models. As it could be seen in case of both the aeroelastic wind tunnel model and the pedestrian bridge, the 3D coupled technique, the modal flutter derivatives method and the quasi-three-dimensional approach provide critical wind speed values close to each other. The main reason for that sort of coincidence is the simplicity of the considered cases in terms of coupling of the physics. If the (atmospheric) turbulence were to be taken into account, the 3D coupled method and the modal flutter derivatives method would be expected to give more accurate results compared to the quasi three-dimensional one. In order to ascertain about that, an advanced wind tunnel test program would be desired to perform, in which the inflow fluctuations representing the atmospheric turbulence should be included. As to the numerical simulation, taking turbulence into account would extremely intensify the computational cost, calling for a more powerful PC or rather clusters. By further developing the introduced 3D numerical approaches, special and unusual structures could be investigated in more detailed and more accurately.

Publications in connection with the dissertation

The relevant publications are enlisted below:

International journal papers:

[1] Györgyi J., Szabó G.: "Dynamic analysis of wind effect by using the artificial wind function", *Slovak Journal of Civil Engineering*, XVI:(3) pp. 21-33, (2008)

[2] Pálossy M., Szabó G., Szecsányi L.: "Mayfly footbridge, Szolnok - design, construction and dynamic behaviour of the longest footbridge in Hungary", *Steel Construction.* 3/2011, (2011)

[3] Szabó G., Györgyi J., Kristóf G.: "Advanced flutter simulation of flexible bridge decks", *Coupled Systems Mechanics*,, Vol. 1, Num. 2, pp. 1-22, (2012)

Hungarian journal papers:

[4] Györgyi J., Szabó G.: "Szélparaméterek numerikus vizsgálata", *Építés és Építészettudomány*, 38:(3-4) pp. 297-328., (2010)

[5] Fornay Cs., Nagy A., Szabó G.: "Az M43-as autópályán épült Móra Ferenc Tisza-híd tervezése", *Vasbetonépítés*, 2011/5, 4. part, (2011)

[6] Szabó G., Györgyi J.: "Numerical simulation of the flutter performance of different generic bridge cross sections", *Periodica Poletechnica-Civil Engineering*, 55:(1,2) pp. 137-146. (2011) IF: 0.455, WoS link, Scopus link, DOI: 10.3311/pp.ci.2011-2.06, (2011)

International conference proceedings:

[7] Györgyi J., Szabó G.: "Calculation of wind effect by dynamic analysis using the artical wind function", 6 Int. Conf. on New Trends in Statics and Dynamics of Buidings, pp. 9-12, #paper, pp. 1-14, Bratislava, 2007. 10. 18-19, ISBN 978-80-227-2732-7, (2007)

[8] Györgyi J., Szabó G.: "Dynamic calculation of reinforced concrete chimneys for wind effect using the different codes and analysing the soil-structure interaction", #paper 1371, pp.1-12, ECCOMAS Thematic Conference on Computational Methods in Structural Dynamics and Earthquake Engineering, Rethymno, Crete, Greece, 2007. 06. 13-16, ISBN 978-960-254-682-6, (2007)

[9] Györgyi J., Szabó G.: "Calculation of dynamic interaction of a train and an arch bridge". In: Frangopol K., (editor) Bridge Maintenance, Safety, Management, Health Monotoring and Informatics, Seoul, South-Korea, 2008.07.13-2008.07.17., pp. 2398-2405. Paper CD. (ISBN: 978 0 415 46844 2), (2008)

[10] Szabó G., Györgyi J.: "Application of the Fluent software in civil engineering", Int. Conf. 70 Years of FCE STU, Section 02, Sub-section A, #paper3, pp.1-17, Bratislava, 2008. 12. 4-5, ISBN 978-80-227-2979-6, (2008)

[11] Györgyi J., Szabó G.: "Fluid-structure interaction analysis with the ANSYS software in bridge aeroelasticity", In: 5th European & African Conf. on Wind Engineering. Florence, Italy, 2009.07.19-2009.07.23. Florance: pp. 1-12. Paper CD 087., (2009)

[12] Szabó G., Györgyi J.: "Three-dimensional Fluid-Structure Interaction Analysis for Bridge Aeroelasticity", WORLD Congress on Engineering and Computer Science 2009, WCECS 2009, Vol II ,#paper, pp. 892-897, San Francisco, USA, 2009, ISBN 978-988-18210-2-7, (2009)

[13] Szabó G., Györgyi J.: "Application of the ANSYS Software to Fluid-Structure Interaction Analysis", The Seventh Int.Conf. on Engineering Computational Technology, #paper 159, pp.1-19, Valencia, 2010, ISBN 978-1-905088-40-9nd, (2010)

[14] Szabó G., Kristóf G.: "Three-dimensional numerical flutter simulation", The Fifth International Symposium on Computational Wind Engineering (CWE2010) Chapel Hill, North Carolina, USA May 23-27, 2010, pp. 1-8 (pen drive), (2010)

[15] Szabó G., Györgyi J.: "Flutter Simulation and Measurement of generic Bidge Deck Sections", In: 9TH Int. Conf. on New Trends in Statics and Dynamics of Buidings. Bratislava, Slovakia, 2011.10.20-2011.10.21. Bratislava: pp. 1-18. (CD) Paper 42., (2011)

[16] Szabó G., Pálossy M., Szecsányi L.: "Three-dimensional forced oscillation technique in flutter assessment", The Seventh International Colloquium on Bluff Body Aerodynamics and Applications (BBAA7) Shanghai, China; September 2-6, 2012, pp. 1-10, (2012)

Other publications in connection with the field of fluid dynamics:

[17] Györgyi J., Szabó G.: "Dynamic calculation of train-bridge interaction at arch bridge", #paper A-0262, pp.1-8, IABSE Symposium on Responding to Tomorrow's Challenges in Structural Engineering, Budapest, 2006. 09. 13-15, (2006)

[18] Györgyi J., Szabó G.: "Dynamic train-bridge interaction at arc bridge", 5th Int. Conf on New Trends in Statics and Dynamics of Buidings, pp. 99-102, #paper, pp.1-14, Bratislava, 2006.10. 19-20, ISBN 80-227-2479-3, (2006)

Presentations:

[19] Szabó G., Györgyi J.: "Szél dinamikai hatásának vizsgálata generált szélsebesség-függvény alkalmazásával", X. Hungarian Mechanics Conference, Miskolc, (2007)

[20] Szabó G.: "ANSYS szoftver alkalmazási lehetőségei az építőmérnöki tervezői gyakorlatban", ANSYS conference and meeting (organized by CFD.hu Ltd.), Budapest, (2008)

[21] Szabó G.: "Nemlineáris- és dinamikai feladatok Ansys környezetben", Gépészet 2008 konferencia, Budapest, (2008)

[22] Szabó G.: "Bridge aero-elasticity simulation by using the ANSYS software", ERCOFTAC Spring Festival „panta rhei” Budapest University of Technology, Budapest, Hungary, May 4th, (2009)

[23] Szabó G., Györgyi J.: "Bridge flutter assessment with three-dimensional fluid-structure interaction simulation", 17th Inter-Institute Seminar for Young Researchers, Krakow, (2009)

[24] Szabó G.: "Bridge flutter FSI analysis", ANSYS conference and meeting, Darmstadt, (2009)

[25] Szabó G.: "Three-dimensional numerical flutter simulation", ANSYS conference and meeting (organized by CFD.hu Ltd.), Budapest, (2010)

[26] Szabó G.: "Híd szekciók numerikus áramlás-szimulációja", ANSYS conference and meeting (organized by CFD.hu Ltd.), Budapest, (2011)

[27] Szabó G., Györgyi J., Kristóf G., Szabó Zs., Zelei A.: "Hídpályák belebegésének numerikus és kísérleti vizsgálata", XI. Hungarian Mechanics Conference, Miskolc, (2011)

[28] Szabó G.: "Három-dimenziós kényszermozgatásos eljárás hídszerkezetek belebegés vizsgálatához", ANSYS conference and meeting (organized by CFD.hu Ltd.), Budapest, (2012)