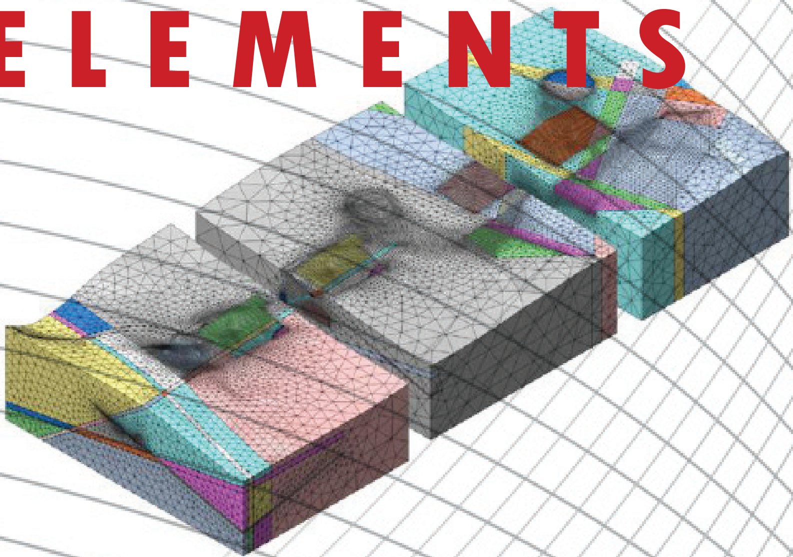
SPRING

2010

DIANA ELEMENTS



TNO DIANA

A TNO Company

Content

Colophon

DIANA ELEMENTS *is published by*
TNO DIANA BV
Delftechpark 19a
2628 XJ Delft
The Netherlands

T +31 (0) 88 24262 00
F +31 (0) 88 34262 99
eFax +31 (0) 84 74990 28

info@tnodiana.com
http://tnodiana.com

DIANA ELEMENTS is distributed to TNO DIANA BV customers and other interested persons.

The editors welcome letters, technical articles, news of forthcoming events, publications of topical interest and project descriptions.

Editorial

Computational Modeling and Analysis goes together with understanding

Business globalization means that engineers are working on more and more international projects. Traditionally, countries have their own engineering methods and codes and in international projects the best elements from each are adopted. Implementation of international projects and international design codes, such as Eurocode, are now changing this practice. For most engineering consultants, despite the economic crisis, innovation of engineering methods is today a hot topic. There is a strong need for tools to evaluate methods and working-procedures. Advanced computational modeling and analysis software can be such tool.

In engineering practice the use of analysis software was for a long time restricted to the design of structures, in which the dimensions of components of structures are checked for strength and deformation with reference to design codes. Now we see more and more that full nonlinear 3-dimensional models are defined because engineers want to understand and optimize working procedures. This trend is not only caused by the globalization and the implementation of Eurocode, but, as I explained in the previous DIANA Elements issue, both hardware (processors and graphics) and software are now so powerful that 3-dimensional nonlinear analysis can be done within a few days.

This development has a down-side and that is that advanced analysis always requires a good understanding of the computational model. When an inappropriate element-mesh is defined, incorrect element types are chosen, boundary conditions are not correct, or the restrictions of a material model are not understood, the analysis results may look fancy but are worthless.

Many have pointed to this risk before. It is always important to build-up the model step by step and to verify every step. Unfortunately we see that this approach is not always followed.

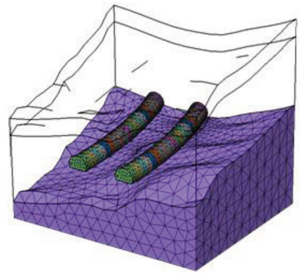
I am very happy that universities now offer more courses in their curriculae in which on-hand training of finite element analysis is provided. TNO DIANA provides training-sessions for the different products on a regular basis and we organize seminars and user-meetings such that users of our software can exchange experiences and learn from each other. In April this year the 4th session of the International Bridge Seminar at Imperial College was organized. And this year, for the first time we combined this with a geotechnical seminar with the title Geotechnical Advances on Urban Renewal. Next month, from 17-18 June the annual International DIANA User-Meeting will take place in Brescia in Italy. The presentations and discussions will not only be of interest for DIANA users but for every engineer who is interested in computational modeling and analysis.

I will be pleased to meet you in Brescia and hope you will enjoy reading this newsletter.

Gerd-Jan Schreppers
Director TNO DIANA BV

International Seminar

Geotechnical Advances in Urban Renewal: Analysis & Design



On the 20th April, the midas GTS team were pleased to join the CIVIL team (see later article) at Imperial College to host the International Seminar Geotechnical Advances in Urban Renewal: Analysis & Design.

This event brought together a combination of professionals and students from the geotechnical and tunnel engineering fields. It focused on current and proposed future practice on analysis and design, paying particular interest in urban renewal topics.

Unfortunately some of the planned presenters were not able to join us from overseas, due to flight restrictions caused by the volcanic eruption. However, at the last moment additional speakers "stepped into the breach" and helped make the event successful.

Our thanks go to those speakers who were able to help out at short notice. On the day presenters included:

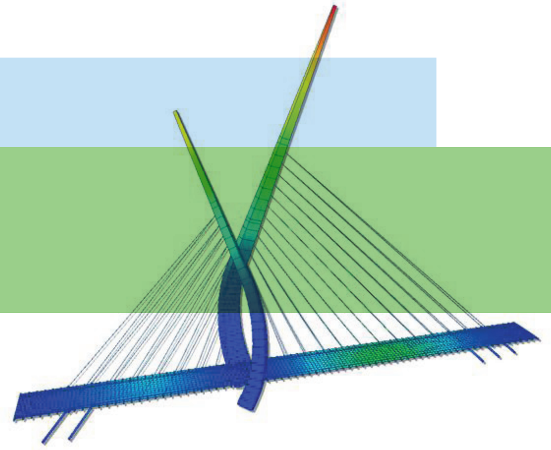
Dr Hoe-Yeow (Arup), Professor Kenichi Soga (University of Cambridge), Mr Alexander Nikolic & Mr Angelo Fasano (Buro Happold), Dr Neil Tsang (Imperial College) and our own Dr Gerd-Jan Schreppers, Dr Pranesh Chatterjee and Dr Ahmed Elkadi.

Available upon request (to info@tnodiana.com) is a proceedings CD, which also includes the originally planned presentations from the speakers below:

- Dr Hoe-Yeow (Arup)
- Professor Kenichi Soga (University of Cambridge)
- Mr Alexander Nikolic & Mr Angelo Fasano (Buro Happold)
- Dr Kurt Zeidler (Gall Zeidler Consultants LLC)
- Professor Fulvio Tonon (University of Texas at Austin)
- Professor George Exadaktylos (Technical University of Crete)
- Dr Neil Tsang (Imperial College)
- Mr Rodney Meadth (Cobalt Construction Company)

4th International Bridge Seminar

Advances in Bridge Design



On the 20th April 2010, the 4th International Bridge Seminar took place at Imperial College London. Despite the grounding of all aircraft across Northern Europe, many bridge engineers were still able to make the journey to London and contributed to the success of the seminar.

There were a total of seven presentations by Chris Hendy (Atkins), Tony Dempsey (Roughan & O'Donovan), Sam Khan (Tony Gee), Professor George England (Imperial College), Mark Palmer (Royal Haskoning), George Daoutis & Desmond Hurley (Hewson) and our own Jurriaan Platenburg. Copies of these presentations can be found on our website.

Much was learned about the advantages of the new eurocodes, the integration of eurocodes in midas CIVIL and the analysis of cable stayed bridges and integral bridges.

Two informative case studies were presented which showed what users of midas CIVIL bridge analysis software are capable of achieving.

Our thanks to those goes to all of the speakers and attendees who, in some cases, went to great lengths to get to the event.

midas CIVIL 2010 - New Release

TNO DIANA BV and midas IT are committed to continually improving the Bridge Analysis and Design software midas CIVIL. Many functions are added to the software every year and improvements are made twice a year based on requests by existing users. In January midas CIVIL 2010 V1.1 was released and in June the software was updated to V2.1.

New functions in midas CIVIL 2010 V1.1 and V2.1 include:

- **Automesh Function**: Easy and fast creation of complex 2D meshes for shell models. Three different mesh generators are available for triangular, triangular/quadrilateral and quadrilateral elements.
- **Dynamic Report Generation**: Generate a report in Microsoft Word including pictures, tables and graphs. Updates and changes to the FE model are automatically reflected in the report.
- **EC2 PSC and Pier Design**: ULS and SLS are checked for prestressed concrete sections. The results are presented graphically, tabular or in a Microsoft Excel design report.
- **Grillage model wizard, Tapered Composite sections, Tendon Profile Viewer, Concurrent reactions for BD37-01 vehicle loads, Pressure type beam load and new materials models including Structural Masonry and Mander model.**

For full details of the all the updates in midas CIVIL 2010 V1.1 and V2.1 please read the release notes available on our website http://tnodiana.com.

Our sales team (sales@tnodiana.com) would be happy to provide you with any additional information, on-site presentations or training. Please visit our website http://tnodiana.com where you will be able to find much more information on midas CIVIL 2010 including tutorials, FAQ, and trial/commercial downloads.

Announcing

7th International DIANA Users Meeting

BOOK NOW
booking deadline expires soon

TNO DIANA BV and the DIANA Users Association will be holding the 7th International DIANA Users Meeting at the University of Brescia, Italy on the 17th and 18th June 2010.

DIANA Users Meetings are held on an annual basis and provide a forum for exchanging experiences with DIANA and for discussing users' needs. The meeting also offers the chance to learn more about current and future developments in the DIANA program.

The two day event will consist of a number of technical presentations , a discussion of users' requests and a social dinner in the beautiful city of Sirmione.

A list of the presentations and information about registration can be found via the announcement on the front page of our website (http://tnodiana.com).

Announcing

3D Finite Element Analysis for Geotechnical & Tunnel Engineering

TNO DIANA BV will be holding the above training course on the 17th and 18th June at its offices in Delft.

The course is free of charge and consists of a balanced mixture of presentations and hands-on analysis with midas GTS finite element software.

The course is aimed at practicing engineers and academics who wish to learn more about 3D finite element geotechnical analysis in general and the software midas GTS.

The content will be relevant to both current and potential users of finite element software and engineers involved in consultancy services.

BOOK NOW
booking deadline expires soon

Further information and registration can be found via the announcement on the front page of our website (http://tnodiana.com)

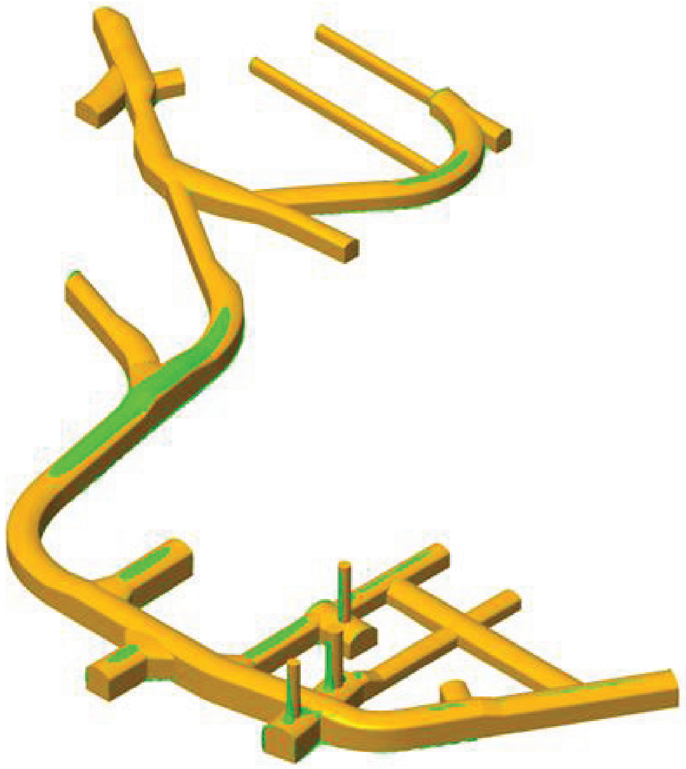
3D Finite Element Modelling in Hard Rock with midas GTS

Authors: Topias Siren, Daniele Martinelli, Antti Sjöblom, Jesse Ström, Rockplan Ltd

Generally one of the biggest challenges in underground construction is the overall stability of the excavations, usually due to the fact that the rock mass is not strong enough. When working with hard rock (intact uniaxial compressive strength 100-300 MPa) some new challenges emerge as some of the old ones just change form. This text covers two different cases in Finland by Rockplan Ltd of utilising 3D finite element modelling in demanding underground hard rock excavation and construction projects. Hard rock excavations in depths up to 500 m possess different kind of challenges than those excavated in the near surface urban areas.

View of a tunnel (photo couresy of Topias Siren, rock engineer at Rockplan Ltd)

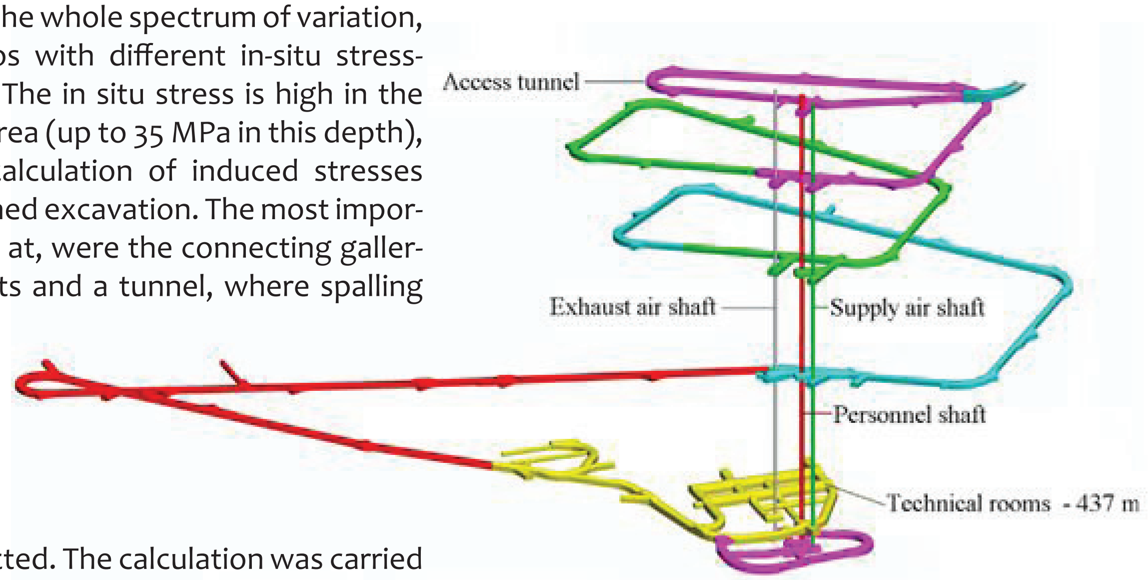
As the mines and nuclear waste disposal facilities expand deeper, the increasing stress magnitudes cause the probability of a failure to increase. The understanding and reliable prediction of the failure process are the key to optimal layout and adequate rock support measures which enable cost-effective and safe construction of a deep underground facility. As the stresses at the excavation boundary exceed the rock mass strength, a brittle failure occurs, that is often referred to as “spalling”. This phenomenon occurs when strong compressive pressure induces crack growth behind an excavated surface and eventually leads to buckling of thin rock slabs. In order to predict spalling, an approximate value of in-situ stress, stress direction and rock mass spalling strength are required. The shape and direction of the tunnel have a drastic effect to the tangential stresses that exist on the tunnel boundary. Therefore a tunnel oriented parallel to the direction of the principal stress, suffers minimum spalling damage compared to a tunnel that is oriented orthogonally to the direction of principal stress. The complex geometry of an excavation affects stress vector directions and boundary stresses which are hard to predict without three-dimensional numerical modelling.



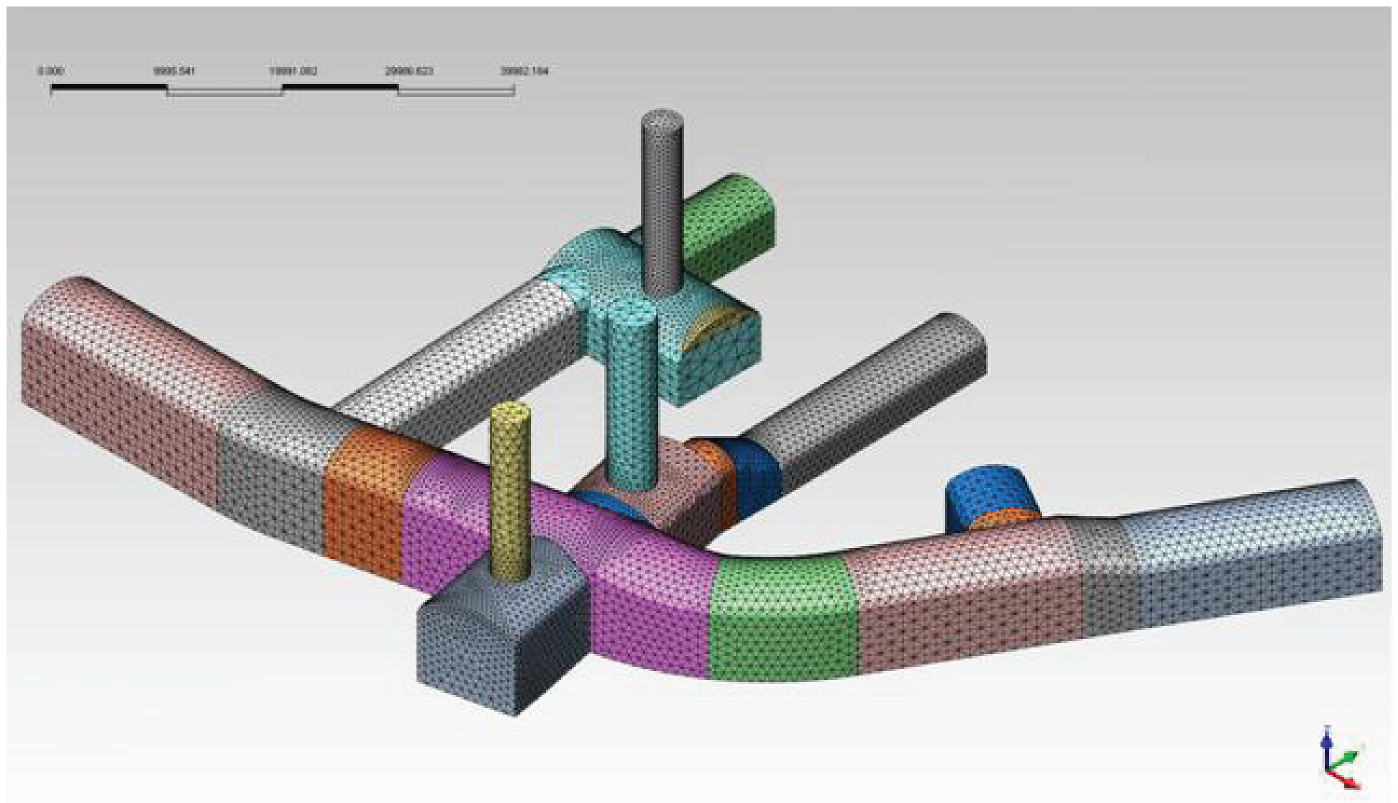
Overview of the areas (green) with spalling risk in the Technical rooms

ONKALO is an underground rock characterisation research facility for future nuclear waste disposal. ONKALO consists of a series of tunnels and shafts starting from the surface and reaching the depth of -460 m. The zone of interest in a large facility was divided into 6 parts. In order to obtain the actual value of stress with high level of accuracy, the order of the elements (around 200,000 per every model) was set to quadratic. An essential problem was to set up the direction and magnitude for the principal stresses.

In order to cover the whole spectrum of variation, different scenarios with different in-situ stresses were created. The in situ stress is high in the Technical Room area (up to 35 MPa in this depth), which required calculation of induced stresses around the designed excavation. The most important parts to look at, were the connecting galleries between shafts and a tunnel, where spalling zones were expected. The calculation was carried out using both elastic models in order to predict the stresses and plastic models in order to predict the deformations. The calculation and interpretation implied that the most critical area will be located at the bottom of a technical shaft that starts from the top of the crown and that will be excavated after the tunnel. This area is a high-stress zone even without the shaft which can be seen as high compression in the arc of the tunnel. After the construction of the shaft, the stress increases up to 120 MPa, which is higher than the compressive strength of the intact rock.



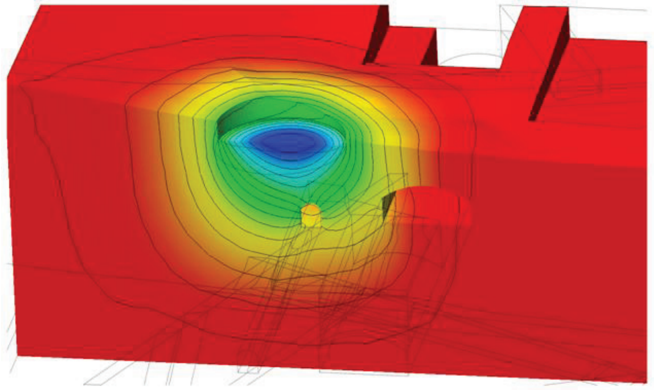
ONKALO preliminary layout (image courtesy of Posiva Oy)



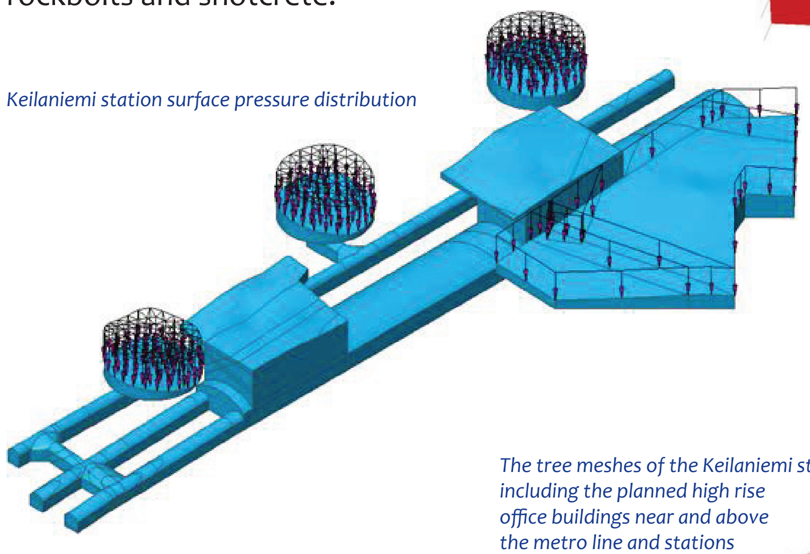
Mesh in the shaft zone

Another tunnel engineering challenge, which is quite common both in soil and different rock types, is the impact of an underground cavern on its surroundings. Vibrations from blasting, fall of the groundwater level and other excavation influenced processes can cause displacements to the structures near the cavern. Due to tectonics the in situ horizontal stresses near the surface in hard rock in Finland commonly vary between 5-15 MPa. In tunnelling projects this usually means rise of the overburden and shear in joints. Uneven rise and settlement alike, can damage foundations. The forces causing the displacements and deformations are high and so usually uncounterable with reasonable reinforcement measures. For that reason additional effort is put on the design of the best shape and placement of the excavation. That is exactly the case in Länsimetro project.

Länsimetro is a new metro line, that is estimated to be open for traffic at the end of 2014, will transport over 100,000 passengers per day from Helsinki centre to Espoo. The project involves around 14 km of tunnels, 7 stations and the removal of around 3 million cubic metres of rock. The tunnels and stations will be excavated by drill&blast and reinforced with rockbolts and shotcrete.

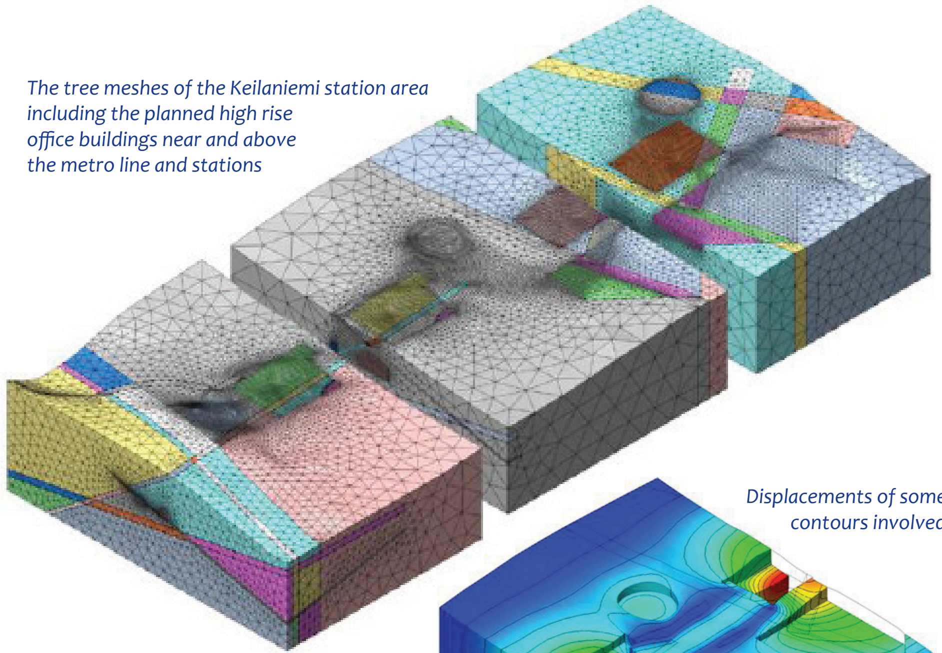


Vertical displacement distribution of a tower foundation

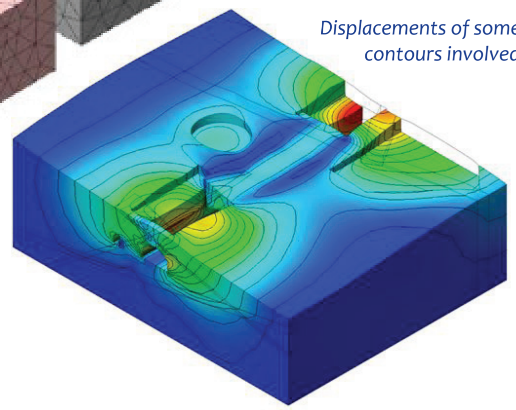


Keilaniemi station surface pressure distribution

A part of the tunnels and the Koivusaari station will be located under the sea. The metro line travels mostly under urban areas with high rise buildings and other critical structures. For example at Keilaniemi there will be four tower buildings 100 m tall, near or above the metro line and station. The foundation pressure of the buildings will be between 2 and 4 MPa each. In situations like this it is vital to forehand simulate the interaction between the tunnels and the surrounding structures so that critical parts of the construction can be identified and the designs can then be optimized accordingly. It has to be confirmed that the additional load on the surface will not damage the tunnels and the tunnels on the other hand will not cause unacceptable displacements on the foundations of the structures above. Working with this kind of cases with such a variable shapes and load distributions 3D modelling is required to reach the required precision.



The tree meshes of the Keilaniemi station area including the planned high rise office buildings near and above the metro line and stations



Displacements of some contours involved

Announcing

midas GTS 2010 - New Release

TNO DIANA BV and MIDAS IT are happy to announce the new release midas GTS 2010 V1.1. This major upgrade is available from the end of June 2010.

This release is special as it reflects a 1.5 years development project where the non-linear solver from the DIANA software has been fully integrated in midas GTS 2010 for nonlinear and seepage, including consolidation, analysis. This proven track record and state-of-the-art nonlinear solver of DIANA brings robustness and versatility into midas GTS.

As part of the integration work circa 500 verification and consistency tests have been defined and analyzed to make sure that all the func-tionality of the integrated program gives correct results.

In addition, the results of circa 200 midas GTS users' models have been analyzed and compared with results of earlier solvers. An automatic testing-system has been defined and put in place, which guarantees that results are checked against target results for every update automatically. We want the user to be able to fully concentrate on the geotechnical problem by using well tested and reliable software.

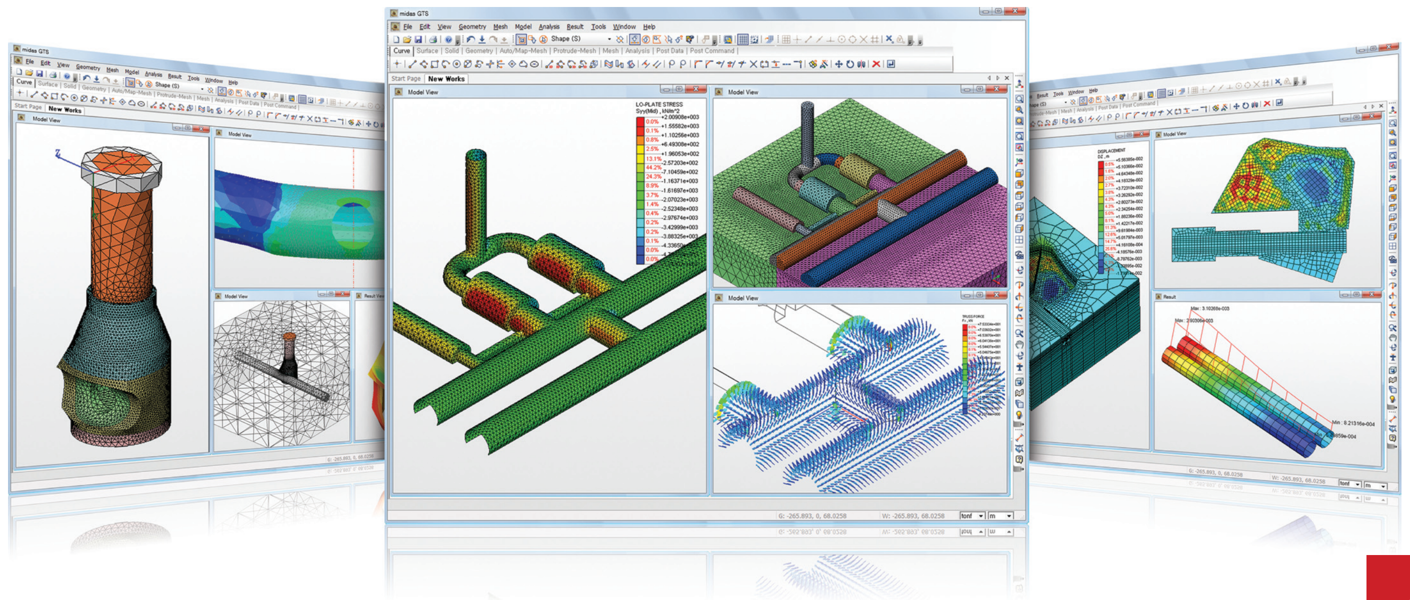
This comes as well with new functions and enhancements based on requests by existing users. Some new functions in midas GTS 2010 V1.1 include:

- Gauging element feature to assist forces in solid elements when used to model structural elements such tunnel linings etc.
- Change of water level per mesh set that is useful in modelling dewatering inside excavations in user-friendly manner.
- Anchor modeling wizard for user-friendly modeling of anchors and rock bolts.
- Support 64-bit OS & multi-core parallel system in nonlinear, construction-stage and seepage analysis.

For full details of the all the updates in midas GTS 2010 V1.1 please read the release notes avail-able on our website http://tnodiana.com.

Our sales team (sales@tnodiana.com) would be happy to provide you with any additional information, on-site presentations or training.

Please visit our website http://tnodiana.com where you will be able to find much more information on midasGTS 2010 including tutorials, FAQ, and trial/commercial downloads.



Ultimate Load Estimation for RC Structures

Authors: Zhanqi Guo, Faculty of Civil Engineering & Geosciences,
Delft University of Technology, The Netherlands.
Gerd-Jan Schreppers & Wijtze Pieter Kikstra, TNO DIANA BV

Calculation of the ultimate load that RC structures can bear is a very important check for design engineers. However, when a RC structure is approaching the ULS state, stress peaks and snap-backs or valleys associated with brittle cracking and subsequent stress redistribution may occur in the structure. Simulation of this process with a non linear finite element analysis not only can be very time consuming, but often requires also advanced solution procedures such as the arc-length method for overcoming numerical instability and divergence of the incremental-iterative procedure. This may discourage design engineers from using non linear finite element analysis. The work presented in this paper is aimed at proposing a methodology alternative to the standard non linear analysis for estimating the ultimate load of RC structures.

Instability of nonlinear analysis induced by occurrence of a negative stiffness matrix can be avoided when a sequentially linear analysis (SLA) is used. Rots [1] and his research group have done much work in this area. Recently de Boer [2] has proposed a so-called sequential procedure for analyzing RC structures. In this procedure the load is applied in several increments. For every load-increment a pre-defined number of non-linear analyses is performed. In every nonlinear analysis steel plasticity and concrete crushing are considered as nonlinear aspects, but the cracking of concrete is not. For elements in which the tensile principle stress is beyond the tensile strength the material model for that element is changed to orthotropic elastic material with a lower stiffness in the direction of the maximum tensile stress. The reduction of the Young-modulus in the direction of cracking is defined by a softening curve. After a few repetitions of the nonlinear analysis for a certain load-increment the material-definitions will not change anymore, and the procedure can be repeated for the next load-increment.

The ULS of the structure is reached when the deformations become very large or when no solution can be found for a next load-increment.

The sequentially linear analysis procedure has been implemented in the development version of DIANA.

Two examples are used for verifying this implementation. The first example (Beam c3), is a three-point bending test. The second example (Beam 8), is a four-point bending test. Both tests were performed at the University of Toronto [3,4]. Both tests were conducted in displacement control

In these examples the Hordijk softening model [5] is adopted in tension, and elasto-plastic behaviour is assumed in compression. Fig. 1 shows the comparison among the load displacement diagrams obtained for Beam 8. The results from a standard non-linear analysis performed with a total strain crack model are compared with the results from sequentially linear analysis that were performed varying the number load-increments (4 and 8), and the number of nonlinear analysis performed for each load-increment (2 and 3).

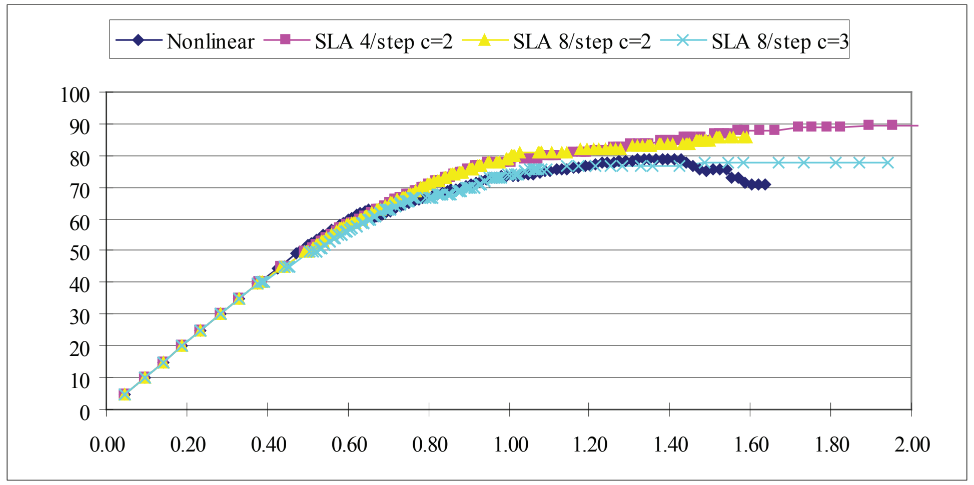


Fig. 1. Load-displacement diagram at the left load point in Fig. 3

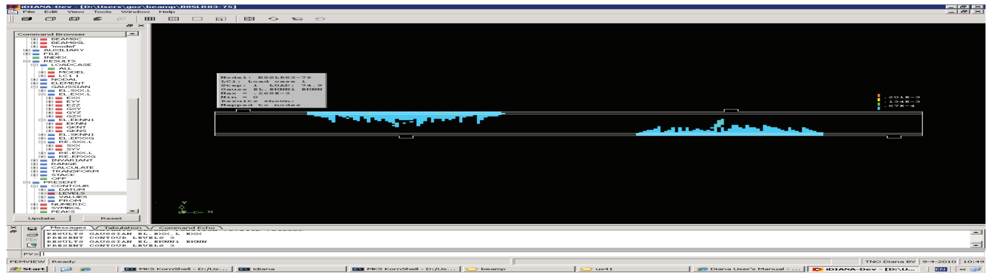


Fig. 2. Principal strain distribution at load level 74.9

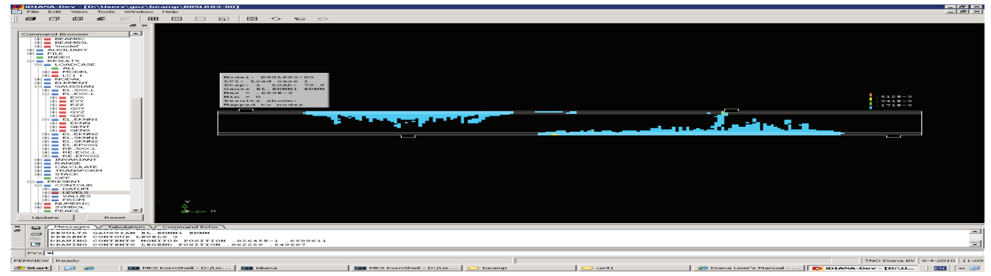


Fig. 3. Principal strain distribution at load level 79,9

Fig.1 shows that a small number of load intrements and analyses per load-increment suffices to guarantee convergence of the analysis to one and the same result as in the non linear analysis, if the ultimate load has not yet been reached. Near the ultimate load, however, convergence is not reached any longer and if the displacement is incremented beyond the ultimate value, the beam exhibits even a residual bearing capacity. In this situation, one can observe that cracks go through the cross section of the beam. Furthermore, in elements that present already cracks, the Young's modulus cannot be reduced more than once. This corresponds to sudden failure of the element, which is more likely to happen near the ultimate load.

In order to reach converging results also near the ultimate load it is necessary to increase the number of analyses per load-increment At failure, however, convergence is not reached.

Based on the results of these numerical tests, the authors suggest that the ultimate load can be defined as the load level for which:

a) the analysis does not convergence
b) cracks go through the cross-section
c) the reinforcement steel yields.

The results obtained from the numerical tests show that the sequentially linear analysis is a very efficient method that allows for a reasonable estimate of the ultimate load and, for this reason, can serve as alternative to the traditional non linear analysis.

References:

[1] Rots JG, Belletti B, Invernizzi S. Robust modeling of RC structures with an "event-by-event" strategy. Engineering Fracture Mechanics 2008; 75: 590–614.

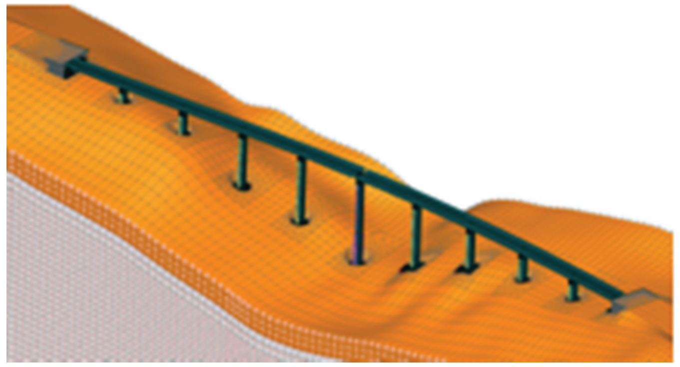
[2] De Boer A. Design strategy structural concrete in 3D – focusing on uniform force results and sequential analysis. PhD thesis, TU Delft, 2010.

[3] Kuchma, D.A., Végh, P., Simionopoulos, K., Stanik, B., and Collins, M. P. The Influence of Concrete Strength, Distribution of Longitudinal Reinforcement, and Member Size, on the Shear Strength of Reinforced Concrete Beams. CEB Bulletin d'Information, 237 (1997), 209-229.

[4] Collins, M. P., and Kuchma, D.A. How Safe Are Our Large, Lightly Reinforced Concrete Beams, Slabs and Footings? ACI Structural Journal, 96-S54 (1999), 473-481.

[5] Hordijk DA. Local approach to fatigue of concrete. PhD thesis, TU Delft, 1991.

Nonlinear soil-structure interaction of a curved bridge located on the Italian Tollway A25



Authors: Mario Ucci , Guido Camata and Enrico Spacone, Engineering Department, University of Chieti-Pescara "G. d'Annunzio"

Preliminary studies

The study was focused on analysis of soil-structure interaction (SSI) in reinforced concrete structures under seismic action.

The first objective of the work consisted in reproducing the effects on the structure of seismic waves amplification due to soil which can cause structural members to undergo critical solicitation regimes. The study was conducted numerically through the direct approach of explicitly modelling a significant part of the soil volume underlying the structure, and including it in time-history analyses, in a more accurate praxis than the bare use of site-specific multiplicative coefficients for the acceleration, which appears to be the current practice. However, such a complex approach, can easily involve computational drawbacks. Therefore, a second objective was to isolate and investigate these issues, in order to define FEM solutions that can best fit the physics of the problem and supply a correct evaluation. In order to do so, several aspects have been considered and subjected to a long series of tests and calibrations (*Fig. 1*), which led to the development of a generalized framework.

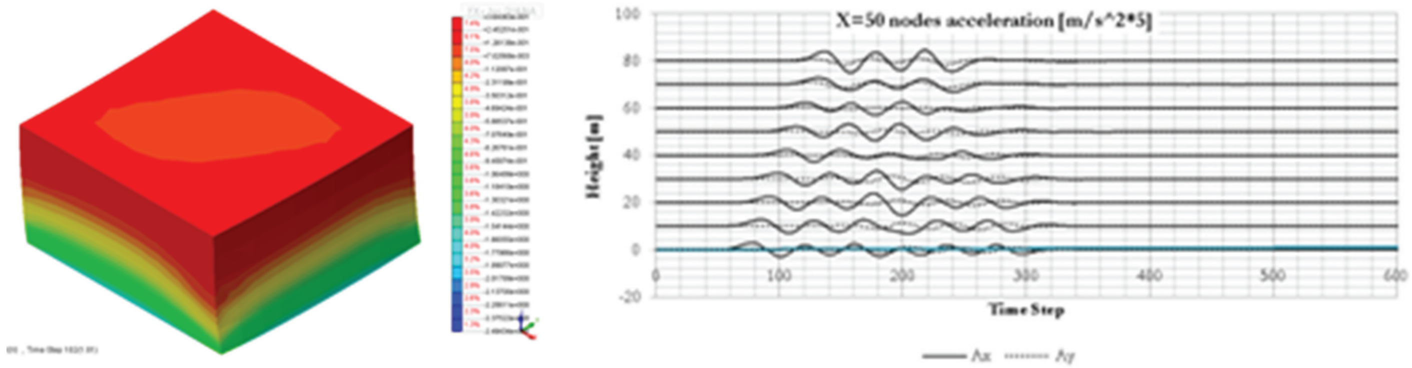


Figure 1. Absorbing BCs calibration tests

One major problem to be fixed, as previously highlighted by many studies[1], comes from spurious wave reflection at the boundaries, which act as rigid limits. Different ways to solve this, correctly reproducing radiation damping, have been considered, but the one giving the best results was the insertion of Spring/dashpot elements placed along the bottom and lateral boundaries of the model; damping coefficients are set distinctly at every node, accordingly with the Lysmer and Kuhlemeyer[2] BC treatment, as follows:

$$\begin{cases} c_p = aA_n\rho V_p \\ c_s = bA_n\rho V_s \end{cases}$$

where A_n is the area of influence of the node, ρ, V_s and V_p are, respectively, the mass density, the shear wave velocity and the compressive wave velocity of the material. a and b are two dimensionless parameters regulating the ratio of incoming energy absorption.

As we can see in Fig. 2, after the setting of absorbing BCs, wave reflection at the boundaries due to superficial mass is no longer detectable.

Another basic requirement of SSI studies is to gain likely deformation field in the soil, consistent with the "repeatability" principle, which has been reached adding node-specific stress time-histories obtained by previous analyses. Since dynamic analysis is required, such a special domain reduction strategy had to be adopted in order to correctly estimate the contribution of the remaining surrounding soil[3].

Soil mesh size represented a cutting edge parameter for analysis feasibility, since dealing with excessively dense 3D meshes can heavily affect analysis time. A study on a very narrow soil column, meshed with elements of varying average size, has been able to predict the quality of the response in terms of spectral superficial horizontal acceleration matching the original record. It revealed a noticeable sensibility at frequencies >10Hz for a meshsize larger than 5m (Fig. 3).

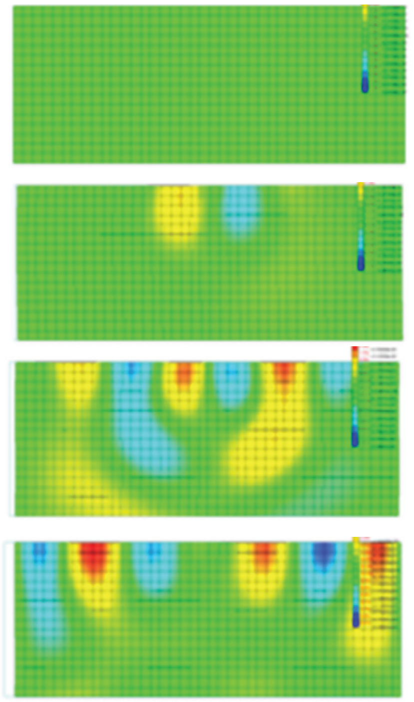


Figure 2. Mass waves absorption

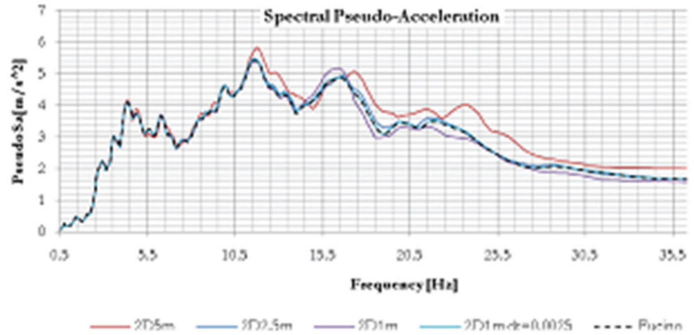


Figure 3. Varying meshize acceleration response

All these tasks have been accomplished even developing *ad-hoc* piece of software (Fig. 4), to be used in conjunction with FX+ and MeshEditor.

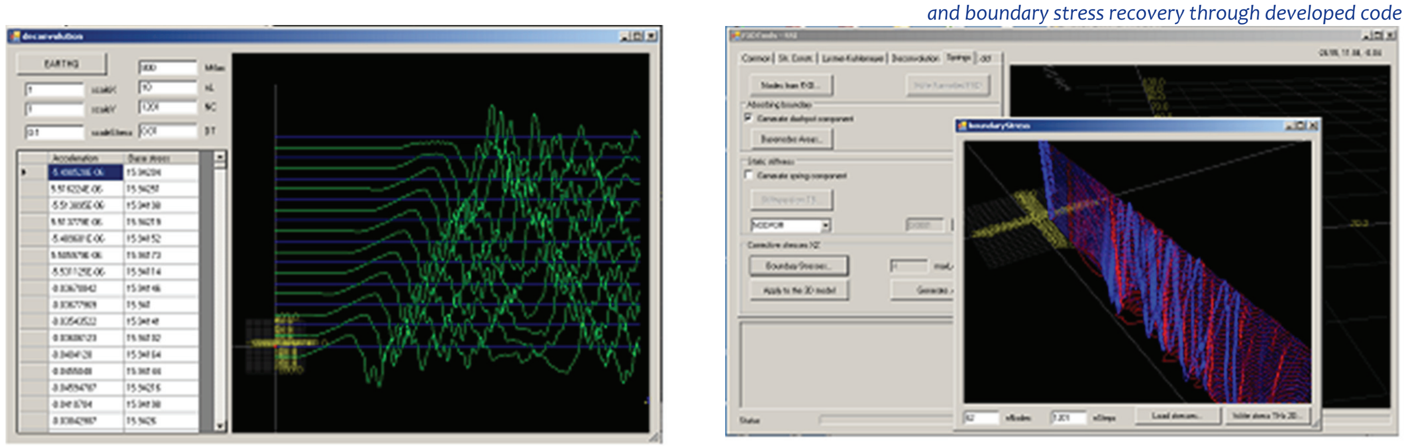


Figure 4. Input deconvolution and boundary stress recovery through developed code

The Bridge

All the investigations are used to calibrate a sophisticate 3D finite element model of the existing "Della Valle" bridge, located on the Tollway A25 (Fig. 5). It was built in the 1970's in the Abruzzo Region, Italy. The "Autostrada dei Parchi" company, which manages the Tollway, has provided extensive information on the bridge design and construction details.

The Della Valle bridge is situated at the crossing point of three heights and so it features complex topography which was supposed to heavily affect seismic amplification due to free surface reflection and consequent interferences of seismic

Figure 5. The "Della Valle" Bridge

waves. Therefore, an adequate description of the free surfaces was required. This level of accuracy was reached by tracing NURBS surfaces based on level curves provided by maps. 3D soil profile is based on the interpolation of surveys conducted in advance to the edification, which revealed the presence of an underlying rock mass with the superposition of a layer of silt pierced by embedded loose concrete fills connecting to foundations.

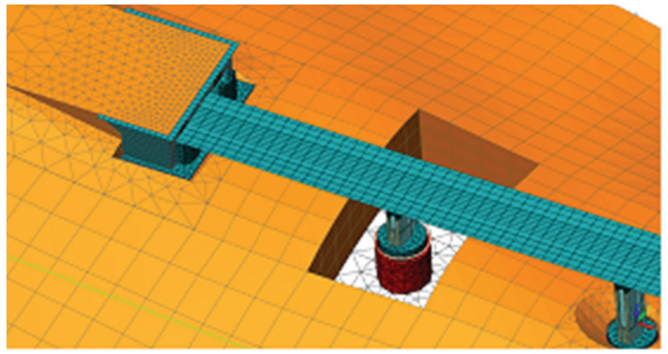


Figure 6. Mesh transitions at the foundation level

Free field areas of the rock and silt layers are treated as structured meshes with brick elements (HX24L), while soil surrounding foundations is treated as transitional areas meshed with tetra elements (TE12L) of decreasing meshsize up to the level required by the superstructures, as shown in Fig. 6. The main soil model spans a 640x130m base reaching pretty differentiated heights, and hosting around 200.000 solid elements.

The bridge is modelled using mathematical surfaces and solids drawn using the pre-processor FX+ for Diana.

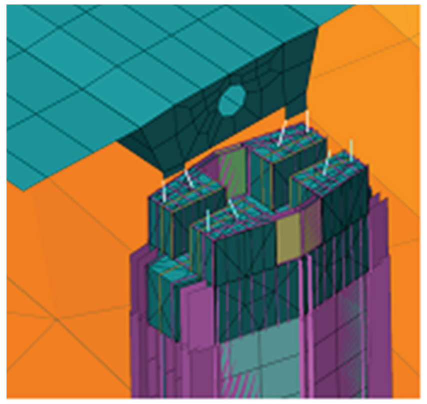


Figure 7. Central hinges

The 392 m long deck has a curved shape in plan and consists of two bridges, one in each direction, formed by continuous pre-stressed reinforced concrete caisson girders, divided at the center by hinges (Fig. 7). The study is conducted on a one-way reduced bridge model. The hollow piers are prismatic and their height varies between 5.5 and 50 meters. Piers walls are treated with linear shell elements with three Simpson integration points over the height. Plinths and heads of the piers are composed by solid elements. Reinforcements are formed by 2D grid layers with equivalent steel areas in the two directions.

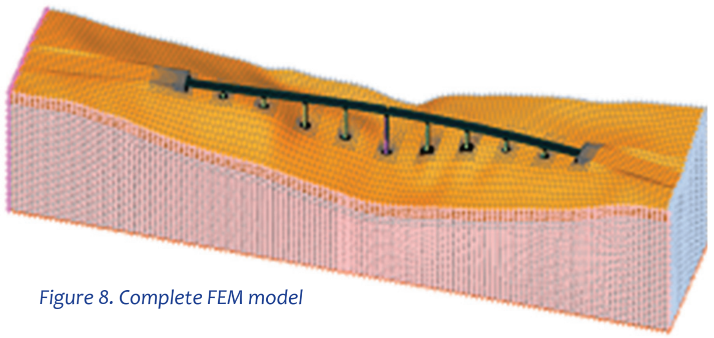


Figure 8. Complete FEM model

Foundations, raising from circular fills embedded in the silt layer are of two kinds: seven circular plinths of variable diameter of 8,50 and 10,00 m and height of 2,00 and 4,00 m and two octagonal tapered plinths at the base of the four central piers. Abutments are characterized by a complex geometry, as they are partly embedded in the silt and raise from rectangular concrete plates, connected to the rock layer through fills. Foundations are modelled explicitly along with the soil so that it is possible to study how the ground motion that is input at the bedrock is filtered at the superficial soil layer, and its effects on the structure.

Realistic nonlinear constitutive models for cyclic loading are used for structural materials. The average concrete cylinder strength (fcm) is assumed to be 25 MPa. The steel is assumed to have an average yield strength of 345 MPa. The complete FEM model is shown in Fig. 8.

The results with fixed base conditions are compared to those obtained including the soil part in the analysis. Results indicated that the bridge response is greatly influenced by the supporting soil.

References:

[1] Conte, J.P., A.Elgamal, Z.Yang, Y.Zhang, G.Acero, and F.Seible (2002), "Nonlinear Seismic Analysis of a Bridge Ground System," (CD-ROM), Proceedings of the 15th ASCE Engineering Mechanics Conference, Columbia University, New York, June 2-5, 2002

[2] Lysmer, J., and R.L. Kuhlemeyer (1969), "Finite Dynamic Model for Infinite Media," Journal of the Engineering Mechanics Division, ASCE, 95(EM4), 859-877.

[3] Proceedings Third UJNR Workshop on Soil-Structure Interaction, March 29-30, 2004, Menlo Park, California, USA

The New Office

On Friday, 26th March TNO DIANA BV moved out of its suite of offices within the TNO building on Schoemakerstraat in Delft.

After a couple of months insitu at the new office, which is located within Delftechpark (still within Delft), we are now proud to call this place "home".

For those of you who have already visited the new office, I think you'll agree that not only is the office in a great location easily reachable from the highway, and links to rail and airports, but also that it has excellent facilities in order for us to host meetings and training events in the future.

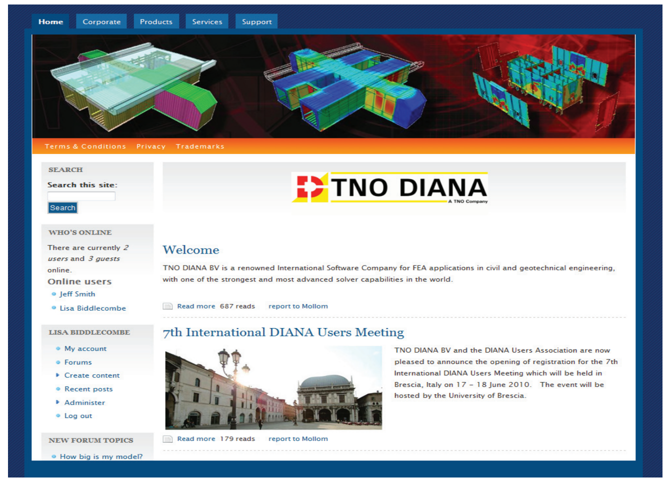
Maps and directions are available on our website.

The New Website

The last six months have been an exciting time for us at TNO DIANA BV, not only have we moved office, but we also launched a new website.

For regular users I doubt you even remember the old website now, but for those of you who have yet to pay the new website a visit, we hope to tempt you with:

- Regularly updated news items such as course announcements, seminar invitations and new releases.
- Product specific areas which provide information about our products and their capabilities
- A library of articles available upon request
- Product specific support pages which include items such as software downloads, patches and manuals, frequently asked questions, tutorials and verification examples.



and

- what we feel most proud of, our user forums which are specific to individual products. Here users can post questions and queries, which are not necessarily just answered by our support team, but also by other users.

Since it was launched the website has continuously evolved and will continue to do so - so don't forget to take a look now and again. We're also always glad to receive any feedback or suggestions - please send these to info@tnodiana.com

http://tnodiana.com

THE TEAM TNO DIANA BV

Gerd-Jan Schreppers
Director

Giovanna Lilliu
Sales Manager
DIANA

Jurriaan Platenburg
Sales Manager
midas CIVIL, GEN & FEA

Ahmed Elkadi
Product Manager
midas GTS

David Begg. Consultant Sales Manager - midas CIVIL, GEN & FEA

SUPPORT & DEVELOPMENT TEAM

Jean-Claude Bauruelle	1st Line Support - DIANA
Kesio Palacio	1st Line Support - DIANA
Chantal Frissen	1st Line Support - DIANA
Hee-Jeong Kang	1st Line Support - MIDAS CIVIL, GEN & FEA
Pranesh Chatterjee	1st Line Support - MIDAS GTS
Jos Jansen	Development Team - DIANA
Pier Nauta	Development Team - DIANA
Jonna Manie	Support / Development Team - DIANA
Wijtze Pieter Kikstra	Support / Development Team - DIANA
Tanvir Rahman	Support / Development Team - DIANA
Arno Wolthers	Support / Development Team - DIANA / ICT

CENTRAL ADMINISTRATIVE SUPPORT TEAM

Jan Kortschot	Sales Support Team Manager
Susann Zink	Sales Support Team
Lisa Biddlecombe	Sales & Marketing Support Team
Jantine van Steenbergen	Office Support

NEW ADDITIONS

Pranesh Chatterjee, Senior Application Engineer - midas GTS

Dr Pranesh Chatterjee completed post-graduation in Geotechnical Engineering and has earned his PhD (Engineering) from Jadavpur University, Kolkata in India working on Seismic SSI analysis. Thereafter, he was engaged as a post-doctoral researcher in the Catholic University of Leuven in Belgium and worked on numerical modeling of train induced vibrations. He was subsequently appointed as Pierse Newman scholar in University College Dublin in Ireland to work on soil-structure interaction problems.

Pranesh has also spent a number of years in the civil engineering software development industry (structural and geotechnical) in the Netherlands, the UK and India. He joined TNO DIANA BV in the beginning of 2010 in the position of Senior Application Engineer and brought in a total of over 12 years experience from academia and industry.

UPCOMING EVENTS IN 2010

Courses, Meetings & Seminars held by TNO DIANA BV

17-18 June	7th International DIANA Users Meeting, Brescia, Italy
17-18 June	GTS Course - Delft, The Netherlands
27 September	GTS Course - Delft, The Netherlands
22 November	GTS Course - Delft, The Netherlands

TNO DIANA expects to be present at the following events:

30 June & 1 July	CDBG, Oxford, England, [CIVIL]
30 Aug - 3 Sept	14th European Conference on Earthquake Engineering, Ohrid, Republic of Macedonia [DIANA]
22-24 Sept	34th IABSE Symposium - Large Structures and Infrasructures for Environmentally Constrained and Urbanised Areas
18 November	Betondag 2010, Rotterdam, The Netherlands [CIVIL/FEA]

To request further information on any of the above events, or to inquire further about any of the enclosed articles please contact us:

TNO DIANA BV
Delftechpark 19a
2628 XJ
Delft
The Netherlands

Tel: +31 88 DIANA 00
Fax: +31 88 DIANA 99

TNO DIANA UK Ltd
Ground Floor - Building 1000
Lakeside North Harbour
Western Road
Portsmouth PO6 3EZ
United Kingdom

Tel: +44 (0) 23 9270 4175
Fax: +44 (0) 23 9270 4001

TNO DIANA North America Inc
38701 7 Mile Road
Suite 260
Livonia
48152-3965
Michigan, United States

Tel: +1 (0) 877 70 34262
Fax: + 1 (0) 320 514 9363

info@tnodiana.com seminar@tnodiana.com courses@tnodiana.com sales@tnodiana.com

http://tnodiana.com