

New in DIANA version 9.3

At a glance:

- Optimized iterative solver
- New direct sparse solver with parallel processing
- 3D line interface element
- Ambient dependent material properties in interface elements
- Modified Maekawa concrete model
- Extended options for dynamic analysis
- Updated version of midas FX+ for DIANA
- Enhanced graphical functionality in iDIANA
- .com extension replace by .dcf extension
-and more!

Iterative solver

The iterative solver is very fast and requires limited memory usage. For this reason it is recommended to use the iterative solver in case of large models containing solid elements. The performance of the iterative solver is further improved in the new DIANA version. A simple example of a footing load test gives a taste of what one can expect from the new iterative solver in DIANA.

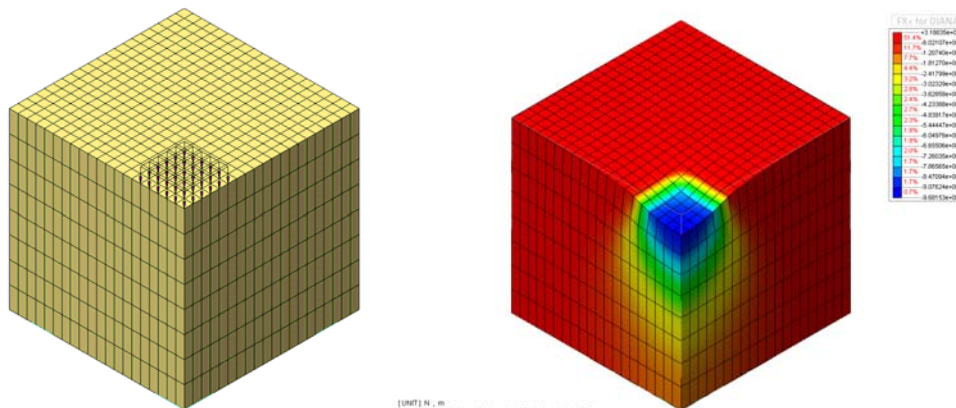


Figure 1: Model of a footing load

The analysis has been performed on a Linux 2.66 GHz Intel® Xeon® X5355 computer with 24 GB internal memory.

Table 1 shows the CPU time for different number of elements and for the different phases of input, model evaluation, solution and output.

Model Name	# HX24L Elements	CPU (s) INPUT	CPU (s) MODEL	CPU(s) SOLVE ITERAT	CPU(s) POST
Model1	20x20x8=3200	0.40	0.98	0.49	0.55
Model2	40x40x16=25600	3.06	8.75	6.87	7.65
Model3	60x60x24=84600	10.21	29.38	32.54	29.11
Model4	90x90x36=291600	34.68	100.09	105.94	97.81
Model5	100x100x50=500000	59.74	182.73	258.31	174.60

Table 1: Performance of the iterative solver in DIANA 9.3

Direct solver with parallel processing

The new direct sparse solver in DIANA 9.3 is the **Intel® PARDISO** solver. This solver can run on multi-processor Intel® processor based Windows or Linux computers. The performance of the Intel® PARDISO is especially good in large models with structural elements such as plates, shells and beams.

An example of performance of the Intel® PARDISO is shown in Table 2, which contains the results from an analysis of a 3D fly-over (see Figure 2).

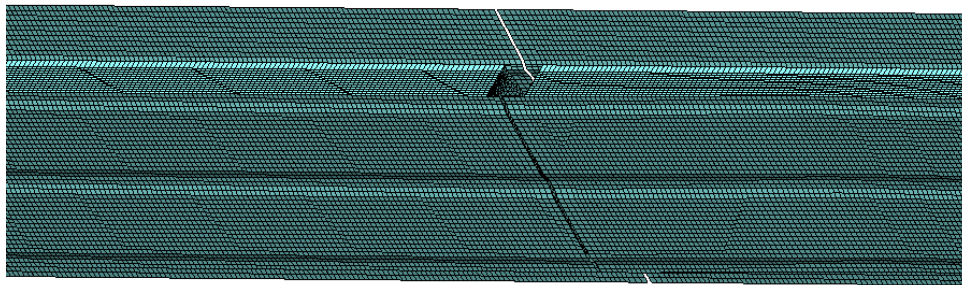


Figure 2: Model of a 3D fly-over

The number of elements in the 3D fly-over is 118.000 (CQ40S quadratic shell elements), with a total of 1.7 million degrees of freedom. The analysis has been performed on a Windows Server 2003 x64 with 2 * Intel® Xeon® X5355 (2.66GHz, quad-core) and 48GB internal memory. The elapsed times for the solution of the set of linear equations are shown in Table 2.

Method	SPARSE SOLVER	PARDISO (1 processor)	PARDISO (2 processors)	PARDISO (4 processors)	PARDISO (8 processors)
Elapsed Time (s)	23 min 0 sec	4 min 36 sec	3 min 4 sec	2 min 30 sec	1 min 54 sec

Table 2: Performance of the Intel® PARDISO solver in DIANA 9.3.

3D line interface element

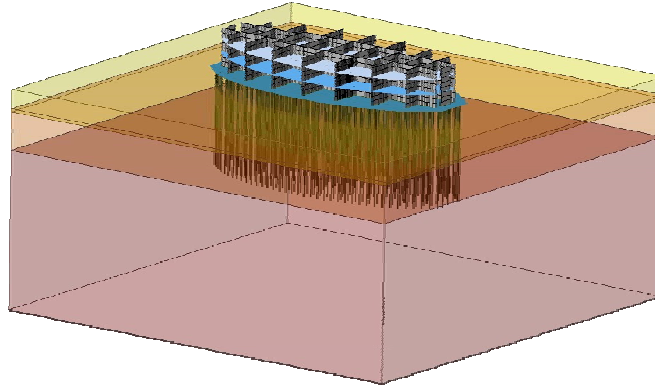


Figure 3: DIANA model of pile foundation in the Shams tower.

The CL18I 3D line interface element is now available. This 3+3 node element can be used for connecting line elements, such as beams and trusses, to solid elements. Typical applications of this type of element are: bond slip of pre-stressed reinforcement tendons in a 3D concrete structure; modeling of frictional interaction between pile foundations and surrounding soil elements. Figure 3 shows a model of the pile foundation of the Shams tower. In this model the new interface element has been applied for modeling the nonlinear interaction between piles and soil.

Ambient dependent material properties in interface elements

The material parameters of interface elements (stiffness, tensile strength, crack-energy and bond slip parameters) can be defined in DIANA 9.3 as function of maturity or temperature. This is especially of interest for analysis of reinforced young hardening concrete, in case primary and secondary cracking occur. Primary cracks can be modeled with interface elements. Secondary cracks can be modeled with a smeared crack model. In the previous versions of DIANA temperature or maturity dependence of material parameters could be taken into account in smeared crack models but not in discrete crack models. Figure 4 shows the model of a thick beam. The reinforcement is positioned at the bottom of the beam. A discrete crack is expected at the left side. The colored lines represent the crack planes of the secondary cracks, which are modeled with a smeared crack model. The color plots indicate the crack-strains.

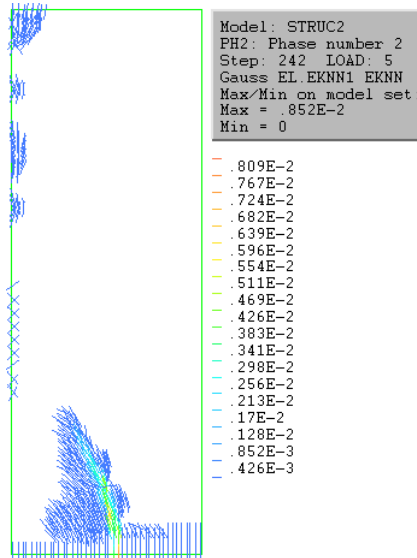


Figure 4: Modeling primary and secondary cracking in a thick wall

Modified Maekawa model

The Modified Maekawa concrete model has been adapted (in consultation with Professor Maekawa). The major change in the model is the switch from the fully 3D elasto-plastic fracture model to the multi-directional uni-axial relations defined for the cracked concrete model, as soon as cracks occur.

Dynamic analysis

Hybrid Frequency Time Domain analysis

A new application for Hybrid Frequency Time Domain analysis (HFTD) is offered in the new DIANA version. This application has been developed for the dynamic analysis of systems in which the dynamic response is nonlinear and frequency dependent. If the dynamic response of any component of the system is nonlinear, analysis in the frequency domain is not applicable, because the response of different excitations can not be superposed. The HFTD procedure is an iterative method that considers nonlinear and frequency dependent behavior of a system by switching between the frequency and time domain. In the HFTD approach a linearized problem is formulated in the frequency domain, while the nonlinear aspects are accounted for in the time-domain. The nonlinear behavior is taken into account by a pseudo-force in the frequency domain. A HFTD analysis in DIANA is a combination of a modal-frequency response analysis and a transient nonlinear analysis and requires the specification of maximum three base excitation loads and corresponding time-load diagrams with equidistant time-intervals. In case of a model containing fluid, structural elements and fluid-structure interface elements, automatically a fluid structure HFTD analysis will be performed. Figure 5 shows the extreme displacements of a seismic analysis of a

concrete gravity dam taking into account the frequency dependent behavior of the reservoir.

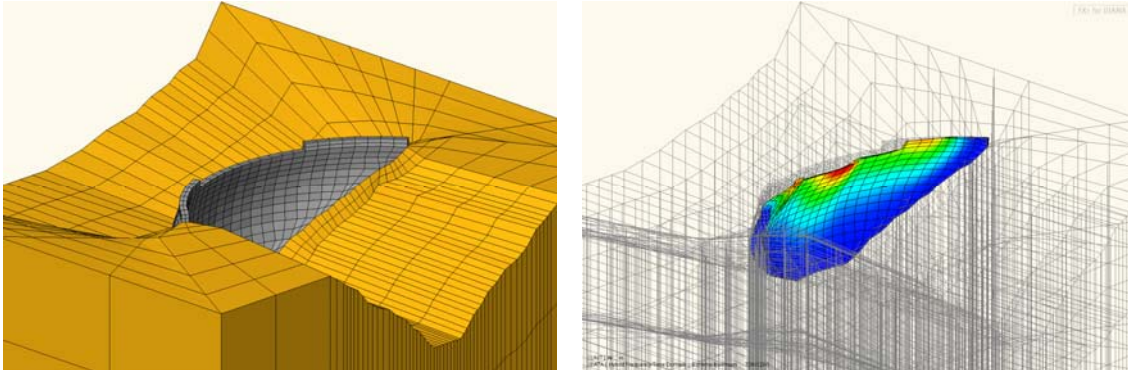


Figure 5: Model of an arch dam and calculation of maximum displacements under assigned earthquake

Dynamic response analysis

The direct frequency response analysis and the modal frequency response analysis procedures have been extended with the possibility to apply multiple loads.

Spectral response analysis has been extended with the possibility to apply multiple base excitation spectra. The frequency dependent load multiplication factors are now specified in the .DAT file (as FREQLO table) instead than in the .COM file.

In modal, spectral, hybrid frequency time domain and eigenvalue analysis the participation factors and (cumulative) effective mass are calculated for each mode, and written to the .OUT file (see Figure 6).

In this manner the user can easier select the desired modes.

MODE	FREQUENCY	GEN.MASS	GEN.STIFFN.	GEN. DAMPING
1	0.37684D+01	0.10000D+01	0.56062D+03	0.86105D+00
2	0.40844D+01	0.10000D+01	0.65861D+03	0.95904D+00
MODE	FREQUENCY	PART.FAC.TX	PART.FAC.TY	PART.FAC.TZ
1	0.37684D+01	0.75349D-08	0.54235D+04	0.24284D-09
2	0.40844D+01	0.66251D+04	-0.24946D-08	0.63003D+03
MODE	FREQUENCY	EFF.MASS TX	PERCENTAGE	CUM.PERCENT.
1	0.37684D+01	0.56774D-16	0.25781D-22	0.25781D-22
2	0.40844D+01	0.43892D+08	0.19931D+02	0.19931D+02
MODE	FREQUENCY	EFF.MASS TY	PERCENTAGE	CUM.PERCENT.
1	0.37684D+01	0.29414D+08	0.13357D+02	0.13357D+02
2	0.40844D+01	0.62233D-17	0.28259D-23	0.13357D+02
MODE	FREQUENCY	EFF.MASS TZ	PERCENTAGE	CUM.PERCENT.
1	0.37684D+01	0.58971D-19	0.26778D-25	0.26778D-25
2	0.40844D+01	0.39693D+06	0.18025D+00	0.18025D+00

Figure 6: Output of participation factors and effective mass

Pre-/Post-processing

Midas FX+ for DIANA

The previous version of midas FX+ for DIANA has been extended with the possibility of:

- process reinforcement stresses and strains in the local reinforcement coordinate system;
- define local coordinate systems for interface elements;
- define nodal sets that are transferred to the MeshEditor and can be used for assigning extra loads or boundary conditions.

iDIANA

The previous version of iDIANA has been extended with the possibility of:

- visualize stresses in a planar development of a curved surface with the command `PRESENT OPTIONS VECTORS MODULATE ZERO`. This is especially useful for analysis of dams. Figure 7 shows compressive principle stresses (in red) and tensile principle stresses (in blue) in an arch dam. The principle stresses are represented as rosettes.

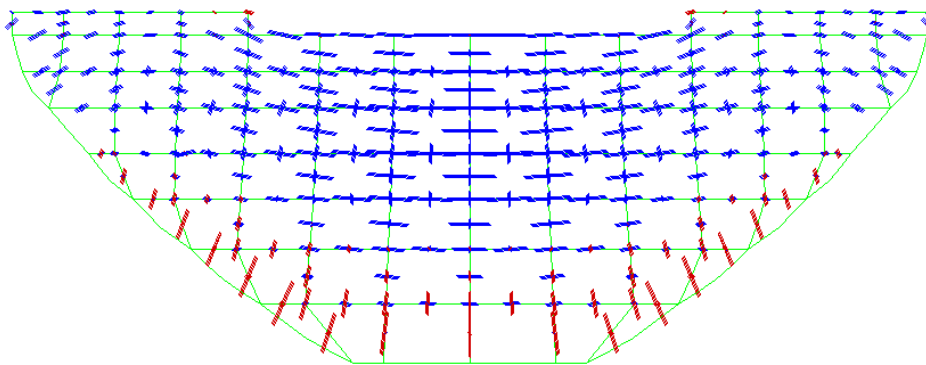


Figure 7: Principal stresses in an arch dam

- visualize geometry properties (thickness of membranes, plates and shell elements, and cross-section of truss and beam elements).
- visualize the crack bandwidth for each element, when a smeared crack-model is used.

These visualization extensions of iDIANA (FEMVIEW) are available for contour plots, numerical values, diagrams, etc.

More news about DIANA 9.3 can be found in the release notes.

If you want further information about this new DIANA version and how you can use it, please contact TNO DIANA at +31-15-2763250 or info@tnodiana.com