# SOLVIA® System 03 – New Features

A number of important new capabilities have been added to the SOLVIA programs and a brief description of the major new features is given below.

## New Parallel Direct Sparse Solver

Storage of system matrices in sparse format is implemented in all SOLVIA versions. In combination with a parallel direct sparse solver very significant speed increases are observed for static as well as dynamic analyses of large models. For example, a test case of a square plate under pressure loading using an 80x80 mesh of 16-node SHELL elements resulting in 287040 degrees of freedom took 22 seconds using the 32-bit version of SOLVIA on an Intel Core 2 Duo processor under Windows XP including calculation of stresses. Frequency, complex harmonic and temperature analyses of large models show also very significant decreases in solution times.

Very large models are solved effectively using the 64-bit SOLVIA versions, either for Windows XP Professional, x64 edition and Windows Vista, 64-bit edition or for Linux x86_64. For example, a 4-node SHELL element model resulting in about 2.5 million degrees of freedom analyzed under 64-bit Windows Vista took 5 minutes in SOLVIA including stress calculations on a PC with 8 GB RAM and an Intel Core 2 Duo processor.

## New Fluid Element

The new fluid element is based on a velocity potential formulation and is intended for static as well as dynamic fluid-structure analysis including both excitations by forces/pressures/structural interface motions and by base motions (earthquakes). The reference static conditions can be analyzed first whereupon a dynamic restart can be performed to add the dynamic effects. Typical applications include earthquake analysis of dams, dynamic loads on fluid tanks and structures submerged in water.

The new fluid element is linear and the adjoining structure may be linear or nonlinear, but with small interface displacements.

Gravity effects on the fluid are considered. The interfaces between the fluid and the structure are established automatically so the new fluid element couples to solid, plane and shell elements without user specification of the interfaces. The fluid free surface is specified by the user and can be loaded by a pressure. A fluid cavity without a free surface can also be analyzed with a specified initial fluid pressure.

In addition to the 6 translational/rotational degrees of freedom that are possible for structural elements, the potential-based fluid element can add the value of the potential and the vertical free surface displacement as extra degrees of freedom at a fluid node as well as the hydrostatic pressure in a connected fluid region.

The fluid pressure and velocity, the fluid free surface vertical displacement and the loading by fluid pressure on the interface to the structure can be displayed by various command functions in SOLVIA-POST such as applicable contour and vector plots, listings, scannings and summations.

The new fluid element is compressible so dilatational waves in the fluid can be analyzed. Surface waves due to gravity can also be considered.

SOLVIA Engineering AB, Trefasgatan 3, SE-721 30 Västerås, Sweden
Phone: +46 21-144050 Fax: +46 21-188890
engineering@solvia.se www.solvia.com

## Large Rotations

Models with arbitrarily large 3D rotations can be analyzed. Director vectors are used to determine the computations of large rotations in RIGIDLINK and in the element types BEAM, ISOBEAM, PIPE and SHELL. These director vectors are updated in each iteration. This procedure ensures that no accumulation of errors can occur for models using only path-independent materials although the 3D rotations may be arbitrarily large.

## Co-rotational BEAM element

The BEAM element with STANDARD, GENERAL, USER or RESULTANT section in large displacement analysis is formulated with a co-rotational element reference system updated in each iteration. The element strains are assumed to be small but the rotations/displacements can be arbitrarily large. The rotations at the element nodes are controlled by director vectors. The RESULTANT section includes a material model where nonlinear moment-curvature and force-strain curves can be specified with isotropic, kinematic or nonlinear-elastic behavior in loading/unloading. The co-rotational BEAM element is very effective which can be important when the number of solution steps is very large.

## User-defined BEAM Sections

Arbitrary BEAM cross sections can be defined by the command SECTION N USER. External and internal section boundaries are then modeled as lines composed of straight and circular segments. The section area and moments of inertia are calculated. The resulting cross section can be plotted separately. SOLVIA-POST can calculate shear stresses at selected stress points using coefficients for the section transverse forces and torsional moment. All cross sections used for BEAM, ISOBEAM and PIPE elements can be plotted at the midpoint locations of the elements in a MESH plot to check the section shape and orientation in the model.

The general section BEAM element allows a non-symmetric cross-section and arbitrary location of the shear center. The number of user-specified stress points is increased to 8. The general section BEAM offers symbolic plotting of the cross-section using a rectangle and a circle with the same principal moments of inertia and area, respectively, as the specified values. Rebars and tendons can also be plotted in the cross-section as for the other cross-section types.

## Stresses in BEAM Sections

An arbitrary section of the BEAM element can be modeled in SOLVIA-PRE using plane elements to obtain stress distributions for cases with Saint-Venant torsion as well as shear in the transverse s- and t-directions. Based on input values of sets of section forces/moments $F_r$, $F_s$, $F_t$, $M_r$, $M_s$ and $M_t$, SOLVIA-POST can then display the corresponding section stresses $\sigma_{rr}$, $\sigma_{rs}$, $\sigma_{rt}$ and the von Mises and Tresca effective stresses as contours over the cross section. Vector plots of the shear stress due to Saint-Venant torsion and transverse shear are also possible as well as listing and scanning operations for all the stresses. Section properties including the shear center location and the warping constant can also be listed.

## Forces from Post-tensioned Tendons

The 3D line geometry of tendons to be used for post-tensioning of concrete structures modeled by BEAM elements can be defined using non-uniform B-splines. One or both of the ends of each tendon can be tensioned in a sequence of applied end forces and length changes from anchorage set. The curvature and wobble frictional losses are calculated and the resulting friction and pressure loads from the tendon as well as the end forces are applied to the BEAM elements.

## Line Diagrams

Nodal and element lines can be plotted in the mesh of a model and the variation of a selected variable along the line in the mesh can now also be displayed. Examples are moment and force variations along beams but any admissible line variable can be selected. A new element line for section forces/moments of SHELL and single layer SOLID elements is introduced.

## Windows Printing and Plotting

Text files and plots can be directed to a printer using the menu of the SOLVIA window. Plots can be written to disk in the EMF format (Enhanced Meta File) in addition to the existing PostScript, HP-GL and other formats.

## Linear Analysis Based on Pre-stressed Configuration

Mode superposition, response spectrum analysis and complex harmonic analysis can be carried out based on linearization of a pre-stressed configuration.

The static contribution from all neglected high frequency modes can be included in response spectrum analysis as well as in the response for each solution step during mode superposition analysis.

## Modeling of Building Processes

The new TIME-ELASTIC material is primarily intended for use in load case analysis where each load case is linear but where load cases may have different material parameters, for example different E-modulus. Time functions may then be used to scale the parameters to their appropriate load case values and all load cases can be analyzed in one run without any restarts.

The TIME-ELASTIC material model may also be used together with the element BIRTH/DEATH option to simulate, for example, a building process where elements are successively added and where the material parameters may be different in the various linear stages.

## Further Details

In addition to the major new features there are also a rather large number of detail improvements in the new SOLVIA® programs.