ANSYS® ASAS™ Version 14.04
ANSYS® AQWA™ Version 5.7A
Femsys™ FEMGV™ Version 7.0-05

Release Notes

November 2006

Century Dynamics Limited
Dynamics House, 86 Hurst Road
Horsham, West Sussex, England, RH12 2DT

Telephone +44 (0) 1403 270066
Fax +44 (0) 1403 270099
E-mail centdyn-uk@ansys.com
URL www.century-dynamics.com

.

Revision Information

The information in this guide applies to all ANSYS, Inc. products released on or after this date, until superseded by a newer version of this guide. This guide replaces individual product installation guides from previous releases.

Copyright and Trademark Information

Disclaimer Notice

U.S. Government Rights

Third-Party Software

The products described in this document contain the following licensed software that requires reproduction of the following notices.

Published in the United Kingdom

ANSYS® ASAS™ Version 14.04, ANSYS® AQWA™ Version 5.7A and Femsys™ FEMGV™ Version 7.0-05 Release Notes

Introduction

During 2005 and 2006 the ASAS and AQWA teams, now part of ANSYS, Inc's overall development resources, have continued to improve capabilities. These improvements are primarily instigated by you the user so that the products continue to satisfy the needs of the communities we supply solutions to. These Release Notes contain a summary of the most significant new features which are introduced with the above releases. This release also includes a number of bug fixes and users are advised to access the Century Dynamics Web site www.century-dynamics.com to obtain details of these.

Another major change to the products involves packaging and licensing. The ASAS and AQWA packages have been completely revised and simplified as part of our integration into the ANSYS organisation. Similarly licensing options have been simplified and the hardware key (dongle) is no longer normally available.

The significant new developments and changes can be summarised as:

Installation and Packaging for All Products (ANSYS AQWA, ANSYS ASAS, Femsys FEMGV)

- Value Priced ANSYS ASAS and ANSYS AQWA packages have been introduced
- In future all licensing will use the FLEXlm® network licensing. Dongle based licences will gradually be transferred to FLEXlm usually at renewal time.

ANSYS ASAS

- Stiffened plate elements introduced
- Extension of Optimised Frontal Solver to Subspace Iteration in ASAS(L)
- All ASAS FORTRAN source code converted to Intel® Visual FORTRAN compiler
- AQWA-WAVE has been extended to transfer AQWA-LINE pressures and motions to ANSYS
- Existing ANSYS users can create ASAS(L) and ASAS(NL) models via an APDL
- Radial gap elements introduced in ASAS(NL) to facilitate tube–in-tube contact typical of offshore conductors
- ASAS-CONCRETE capabilities have been extended to incorporate crack width calculations for strength assessment
- ASAS(NL) is able to accept RAO based motions from AQWA for time history analysis
- Shell New Wave formulation implemented in ASAS(NL)
- Number of Wavelets used to define spectral waves increased to 1000.
- Stiffened Plate modelling capability added to FEMGV
- Section offsets can be defined locally in FEMGV
- For ANSYS users an ANSYS to ASAS translator has been created which enables ASAS models to be created from ANSYS

ANSYS AQWA

- The AQWA to Microsoft® Excel® interface AQL has been extended with more commands
- For ANSYS users an ANSYS to AQWA translator has been created. This creates AQWA-LINE models
- Comments can now be inserted anywhere in a data file
- Number of wave directions has been increased from 10 to 41
- Hull drag directions are specified separately from wave directions

- Power Spectral Densities and significant values of mooring tensions and articulation reactions are printed
- The AQWA-GS mesh generation capabilities have been improved. Transom sterns can now be handled
- Hydrodynamic calculations in AQWA-LINE have been improved
- The NPD wind spectrum has been added
- Maximum model size has been increased significantly
- Maximum number of mooring properties has been increased significantly
- Generation of frequencies and directions has been simplified

Installation and Packaging for All Products (ANSYS® AQWA™, ANSYS® ASAS™, Femsys™ FEMGV™)

Value Priced ANSYS ASAS and ANSYS AQWA packages have been introduced

With the integration into ANSYS, the AQWA and ASAS products are now only available as standard 'Value Packages'. Individual modules such as the BEAMST code check module are no longer available but are bundled within the value packages. Existing value packages such as the ASAS SUPERPACK and AQWA Offshore have been superseded. The new packages are:

ANSYS ASAS containing ASAS(L), ASAS(NL) and BEAMST. Existing ASAS active licensees will therefore gain *ASAS(NL)*
ANSYS ASAS-OFFSHORE containing ASAS above plus ASAS-WAVE/MASS, SPLINTER, FATJACK and WINDSPEC. Existing ASAS-OFFSHORE active licensees therefore gain *ASAS(NL) with coupled wave interaction and WINDSPEC.*
Femsys FEMGV containing IGES™, DXF™, ASAS, ANSYS, ABAQUS® and NASTRAN® interfaces. Existing ASAS-SUPERPACK active licensees therefore gain the additional *BEAMST and ASAS(NL) solvers plus IGES, DXF, ABAQUS, ANSYS and NASTRAN interfaces for FEMGV.*
ANSYS ASAS-CONCRETE is unchanged
ANSYS ASAS-PANEL is unchanged

Joint Flexibility is now included as standard in ANSYS ASAS as is the ability to work with large files within a 32 bit environment as is 64Gb file addressability and high speed solver.

ANSYS AQWA Diffraction containing AQWA-LINE, AQWA-GS and AQWA-WAVE.
ANSYS AQWA-SUITE contains AQWA Diffraction plus AQWA-LIBRIUM, AQWA-FER, AQWA-DRIFT and AQWA-NAUT
ANSYS AQWA Cable Dynamics is an additional feature which adds cable dynamic capability to ANSYS AQWA SUITE

In order to gain from the new packages users must be on annual licences or TECS (Technical Enhancements and Customer Support) for Paid-up licences.

Licensing is now based on FLEXlm® network licensing

Dongle (hardware key) licensing is being phased out. Previously both dongle and network licensing options were available. ANSYS® only uses FLEXlm network licensing, the same system already employed by ANSYS ASAS™, ANSYS AQWA® and Femsys® FEMGV®. During 2006 we have been converting a number of dongle licensees to network and we expect this to continue during the remainder of 2006. Also for existing network licensees we will be changing from FLEXlm 9.5 to FLEXlm 10.8.

Dongles offer the facility to move the license from computer to computer. A ‘borrowing’ feature has been introduced in FLEXlm which permits the temporary transfer of a licence to another computer so that the same flexibility as dongles is available. Network licensing does offer the advantage that any computer on a network can access the licence if it is not being used. This facility also extends to individual modules within a ‘Value Package’.

ANSYS® ASAS™

Stiffened Plate Element

A stiffened panel model has been implemented into ASAS to enhance its modelling capability in this area. In this model, the effective stiffness of the stiffeners are computed and added to the plate stiffness to obtain an equivalent material stiffness matrix for the stiffened panel. This enables coarse mesh to be adopted in the analysis. The capability is particularly suited to modelling ship type structures and similar offshore structures.

- Stiffened panels can be modelled by shell element types QUS4, TCS6 and TCS8. The geometry and material details of the stiffened panel are specified using special material and geometric property data.
- The stiffened shell element must be associated with a LAMI material (c.f composite shell).
- The section profile and properties of the stiffeners are defined in the Section data. The section profile can either be chosen from a built-in type (such as TEE) or defined as a fabricated beam section (FAB). The latter allows for the definition of general section shapes.
- Detailed information of the shell and stiffeners that form the panel (material, section name, orientation, spacing etc) is specified as geometric property data in a way similar to the composite laminate definition.
- The input material and geometric property data of the shell and stiffeners are converted to equivalent anisotropic material stiffness for the stiffened panel in stage 1. During this process, the stiffeners in each stiffening direction are treated as an equivalent plate layer by smearing out the axial, bending (about the shell surface, i.e. section ZZ axis) and transverse shear (transverse to the shell, i.e. section XZ axis) properties of the beams. All the subsequent element calculations are then carried out as if it is a homogeneous anisotropic shell.
- The stress resultants in the shell and the stiffeners can be separately computed utilizing the strain resultant results of the stiffened panel together with the ‘layer’ material and geometric properties. This calculation is implemented in ASAS and the results are saved to the results database under result type ‘LAYER STRESS RLT’. If RESU is specified, the layer stress resultant computation will be performed by default but with no printing to the output file. This calculation can be disabled using option NOSS. Also, printing of the layer stress resultants can be activated by specifying option PSHS.

Extension of Optimised Frontal Solver to Subspace Iteration in ASAS (L)

An optimised frontal solver was implemented into ASAS some time ago. For a wide range of problems, the new solver has proved to be superior to the old solvers available in ASAS in terms of both solution time and storage requirement. Use of the solver, however, has been limited to static analyses only.

The solver has been extended to include natural frequency analyses using the subspace iteration technique (SPIT). The availability of the new solver with SPIT can help to reduce the time spent on the equation solving phase of the algorithm. This saving can be significant for large models because the system equations are solved repetitively to determine the natural frequencies.

All ASAS FORTRAN converted to Intel® Visual FORTRAN compiler
For a number of years ASAS has used the Compaq® Visual FORTRAN complier. This is no longer supported and so all the source code has been recompiled using the Intel Visual FORTRAN compiler. Existing users who use ASAS tools to interface to ASAS™ may in certain circumstances have to convert their FORTRAN programs to this compiler, but in most cases this should not be necessary.

AQWA-WAVE has been extended to transfer AQWA-LINE pressures and motions to ANSYS®

AQWA-WAVE is an interface program that calculates and transfers the hydrodynamic loads obtained from an AQWA-LINE run to ASAS for stress analysis. This program has been extended so that for ANSYS users pressures and motions calculated in AQWA-LINE can be transferred to an ANSYS model.

Existing ANSYS users can create full ASAS(L) and ASAS(NL) models via an ANSYS APDL translator

The ANSYS to ASAS translator uses the ANSYS APDL macro language (ANSYS Parametric Design Language). The main functionalities and limitations of this macro are described as follows:

- The translator can translate the nodal coordinates, element topology and suppression data properly. It can handle solid, shell or beam meshes. For ANSYS elements with collapsed edges, they are converted to the relevant triangular (or wedge) elements in ASAS

- The ANSYS element type reference number (that defined in ET command) is converted to an ASAS group number. This is useful since the group definitions are used in some of the AQWA-WAVE data.

- Material property data are converted assuming that they are temperature independent linear elastic isotropic. Note that the material properties specified in the ASAS data will not be used by AQWA-WAVE and hence the actual values are not important.

- Geometric property data are transferred. For shell elements, the thickness is assumed to be constant. As for the material property data, the geometric property data will also be unused in AQWA-WAVE.

- The AQWA-WAVE data requires definition of the wetted surface (i.e. which side is the outer surface for shells and solids). The ANSYS model must therefore contain a dummy pressure load case that defines the wetted surfaces. This loading will be converted to pressure load case 1000 in the ASAS input load data.

- The macro can be executed in ANSYS by entering the command:

  ANSTOASAS,'*filename*',ASASkey

  where ANSTOASAS is the command name, *filename* is the output ASAS data file name (a .asas extension will be appended for ASAS(L) and .asnl for ASAS(NL)) and ASASkey is the translate option (0 to generate data for AQWA-WAVE (default), 1 for full ASAS analysis, 2 for ASAS(NL) analysis). Note also that *filename* must be given in quotes.

Radial gap elements introduced in ASAS(NL) to facilitate tube–in-tube contact typical of offshore conductors

Concentric tubes are typical in offshore structures, e.g. conductors; this new facility in ASAS introduces an ability to model gapping and contact between these tubes. A radial gap element, GAPR, has been added to ASAS(NL) which can be used to model tube-in-tube contact as well as external contact between two parallel tubes.

The formulation of the radial gap element is based on that of the GAP2 element. Thus, it is a 2-node element with 3 degrees of freedoms per node (X, Y and Z). Contact between the nodes is modelled using the penalty method.

ASAS-CONCRETE capabilities have been extended to incorporate crack width calculations for strength assessment

ASAS-CONCRETE provides a post-processing function to ASAS by accessing the ASAS(L) results file and using the results to perform strength, serviceability and fatigue assessment for specific parts of the structure. The structures need to be essentially slab-like and are usually pre-stressed and reinforced. A local non-linear analysis is performed to determine the distribution of forces in a section upon which the checks to various codes are performed. In order to meet increased rigour in the assessment improved tri-axial crack width calculations have been introduced.

ASAS(NL) is able to accept RAO based motions from AQWA™ for time history analysis

The response of a floating body to waves is typically described by the Response Amplitude Operators (RAOs) at a reference point on the body. For each freedom, the displacement RAO consists of two numbers (amplitude and phase or real and imaginary components) that define the vessel response to one particular wave direction and frequency. RAOs can be obtained either from specialist hydrodynamics software or model testing. For instance, AQWA-LINE always computes and stores the RAOs at the centre of gravity of the body.

The time history motion of a vessel can be determined from the RAOs for a given sea spectrum. This time -history can then be applied to define the motion of a point on the structural model in a transient analysis. A typical application is a riser analysis where the tip of the riser is attached to a vessel and hence its motion can be deduced from the RAOs. The advantage of utilising RAOs to calculate motion is that the analysis can be repeated with a different spectrum without the need to create a new time history for the motion of the attachment point.

A facility in ASAS(NL) has been added to accept the RAO data computed from AQWA so that they can be used in combination with the wave load data in a transient analysis.

Shell New Wave formulation implemented in ASAS(NL)

The Shell New Wave theory is a special wave theory which is formulated as a ‘dispersive wave’ and has only been available in ASAS-WAVE but not in ASAS(NL). This wave formulation has now been added to ASAS(NL) thus permitting transient analysis using the Shell New Wave.

Maximum Number of Wavelets

The maximum number of wavelets in an irregular wave has been increased from 200 to 1000. This permits more complex wave spectra to be defined as required in certain areas of the world.

Femsys™ FEMGV™

Stiffened Plate modelling capability added to FEMGV

Further to the stiffened panel development detailed above additions to the FEMGV user interface have been introduced to give the user the ability to create ASAS™ input data files containing stiffened panel information. In addition stiffened plate analysis results can by displayed by FEMGV. The approach uses the composites capability which has been in ASAS for some time to differentiate between the different parts which constitute the stiffened plate.

Section offsets can be defined locally in FEMGV

Individual stiffener elements section offsets can now be defined within FEMGV.

ANSYS® AQWA™

The AQWA to Microsoft® Excel® interface AQL has been extended with more commands

Some new functions have been added to AQL, the interface between AQWA and Microsoft Excel:

Function	Value returned
aqlcogcoord	CG position in structure
aqlnstructs	Number of structures in model
aqlnfreqs	Number of wave frequencies for structure
aqlndirns	Number of wave directions for structure
aqlwavedirn	Wave direction corresponding to wave number
aqlstatposcog	Position of CG after AQWA-LIBRIUM run
aqlstatposnod	Position of node after AQWA-LIBRIUM run
aqlrao2	Interpolated RAOs, given direction and frequency

For ANSYS users an ANSYS to AQWA translator has been created using ANSYS APDL.

An ANSYS APDL Macro has been developed that writes an AQWA-LINE data file from an ANSYS model. This allows all the model generation capabilities of ANSYS to be used to create an AQWA model, including CAD interfaces and advanced meshing.

ANSYS users who have the preview version of ANSYS 11.0 will have a preliminary version of the macro "anstoaqwa.mac". With this release we are including an updated version that is installed in the \aqwa\utils directory. If you wish to use it you should copy the file anstoaqwa.mac to the \apdl directory in your ANSYS installation. The translator should work with any version of ANSYS. Instructions for use are given in the file \aqwa\docs\ansys_to_aqwa.pdf. *Note that since the macro is in an editable text form that the user can modify, Century Dynamics cannot be responsible for supporting the macro as part of AQWA Support Services. The same will apply to the ASAS macro.*

Comments can now be inserted anywhere in a datafile

It is now possible to put comments anywhere in a data file. Comments must start with a * in column 1.

Number of wave directions has been increased from 10 to 41

The limit of 10 wave directions has been a restriction in AQWA-LINE for some time. This limit has now been raised to 41 directions when there is no symmetry. With one axis of symmetry (SYMX or SYMY) the limit is 21 directions, and with two axes it is 11. This allows wave directions to be spaced at 9° intervals, or 10° plus the quartering directions.

Hull drag directions are specified separately from wave directions

It is recognised that it would be inconvenient to have to input hull drag coefficients (CUFX in Deck 10) for all 41 directions, so the definition of hull drag has been separated from the AQWA-LINE wave directions. The directions for hull drag coefficients can now be specified with new cards DIRN, SYMX, SYMY in deck 10. See the reference manual for further details. The default is to use the AQWA-LINE directions, so existing files will still run.

Power Spectral Densities (PSD) and significant values of mooring tensions and articulation reactions are printed

The PSD and significant values of mooring tensions and articulation reactions can now be printed in the .LIS file. This is switched on with a new option 'PRTS'.

The AQWA-GS mesh generation capabilities have been improved. Transom sterns can now be handled

The mesh generator in the AGS could not close a transom stern, which meant that the .LIN file always had to start and finish with a line at y=0. In some cases this could lead to a poor mesh. The mesh generator has now been improved so that it will close the ends of the vessel with a flat surface at the stern or the bow.

Hydrodynamic calculations in AQWA-LINE have been improved

The accuracy of the Green's function and waterline integrals has been improved. This leads to better near-field drift coefficients that now agree more closely with the far-field coefficients. Efficiency improvements have also been implemented which should improve performance considerably for diffraction/radiation simulations.

The NPD wind spectrum has been added

In addition to the three existing types of wind spectra it is now possible to use the NPD wind spectrum, as recommended by the A.P.I. This is achieved by using a new NPDW card in Deck 13.

Maximum model size has been increased significantly

The maximum number of defined nodes has been raised to 15000. The maximum number of defined elements has been raised to 12000, with 8000 of these diffracting. These limits are increased by a factor of two if one symmetry card (SYMX or SYMY) is used, and by a factor of four if both are used. Note that the diffraction calculation for a model with 8000 diffracting elements will take a long time.

Increased model size means larger node numbers, and it was felt that the node offset method is too cumbersome. A new NOD5 card has been added to simplify the input of 5-digit node numbers.

Maximum number of mooring properties has been increased significantly

The maximum number of non-linear mooring properties (CATN, COMP, ECAT, LE2D, POLY, SWIR cards) has been raised to 1000. The maximum number of composite line databases (sets of data headed by COMP or LE2D cards) has been raised to 100.

Generation of frequencies and directions has been simplified

AQWA can now generate sets of frequencies and directions using modified FREQ and DIRN cards in Deck 6. This should make data preparation a little easier for the large numbers of frequencies and directions that some users are starting to include.