Release Notes – ASAS 14.02 and AQWA 5.5

These release notes cover the next releases of ASAS and AQWA which are available to all *bona fide* Annual Licensees or Paid-up licensees with Support Services and who are entitled to the new releases. We have been extremely busy extending the functionality of both products. In particular we addressed a number of user suggestions for developments which I am sure you will find beneficial.

Summary of New Features

ASAS

- Transient non-linear thermal analysis including forced convection/radiation in ASAS(NL) and transfer of thermal loads to stress analysis.
- Job Replace (overcomes need to use COMPED)
- ASAS2FV enhancements to improve ASAS to FEMGV transfer
- Coupled Wave loading enhancements
- Revised security controls
- Surface stress consistency for shell elements in ASAS(L), ASAS(NL) and FEMGV
- Greater flexibility for plate and shell stress orientation in conjunction with FEMGV
- Hysteretic damping in ASAS(NL)
- Directional pressure loading
- More accurate methodology for ASAS-WAVE pressure calculations

ASAS-VISUALIZER

- Viewing of results by SETS
    - By listing
    - By mouse selection
    - As defined in ASAS data
- Applied load values displayed by vectors and per element
- Automatic mode shape animation
- Extract and display the maximum value across a set of load cases
- Display deflections as numeric, contour and deflected shape
- User defined font sizes, style, position
- User defined Unity Check bands for improved presentation

- Display of beam offsets

AQWA

- Extension of convolution “memory” to 120s (5.4F)
- Increased solution speed (5.4G)
- Import of QTF database (5.4H)
- Addition of Fender moorings (5.4J)
- Reading external forces from user-defined .dll. (5.4K)
- Friction in articulations (5.4L)
- 23 Usability improvements (5.5)
- Improved tether calculations (5.5)

More details of these features can be found within the following text. All user documentation has been modified to correspond to these new features. ASAS users will note that we have expanded the description of the new features because we have found that changes to the user manuals do not always communicate the availability of new features sufficiently.

ASAS 14.02

Transient Thermal Analysis

Some years ago ASAS-HEAT ceased to be supported or developed. This was a transient non-linear heat conduction analysis incorporating convection and radiation. A few years ago the steady state non-linear heat conduction capabilities were added to ASAS(NL). With the release of ASAS 14.02 full transient non-linear heat conduction capability is available including forced convection and radiation effects. A new Appendix N in the ASAS(NL) manual gives details of the capabilities.

As with stiffness analysis the transient thermal capability can use the temporal integration scheme.

It is possible to transfer the temperatures as thermal loads to an ASAS linear or non-linear stiffness analysis using a new command GRES. The temperature loads are added to the ASAS database for reading by ASAS(L) or ASAS(NL). Additional thermal loading can also be specified.

AXIAL field elements (linear variation of variable) are used for radiation and convection coefficients.

The following provides a more detailed overview of the analysis characteristics and features of non-linear transient thermal analysis.

Material Models for Convection and Radiation

Up to ASAS 14.02, heat losses due to convection and radiation were modelled indirectly through modifying the conductivity properties. However, the data preparation required to achieve such modelling was sometimes quite tedious. As these heat losses are important elements in heat analysis, two new material models have been implemented into ASAS(NL) to model their effects directly.

a. Convection material

The convective heat transfer per unit area is given by:

$$q = h(T_2 - T_1)^r$$

where h is the film coefficient, r is the power index of the heat transfer law, T_1 and T_2 are temperatures on surfaces 1 and 2.

The new data format of the convection model is:

├── (mat) ── CVEC ── h ── r ── (temp) ──

b. Radiation material

The radiant heat transfer per unit area is given by:

$$q = \sigma\varepsilon(T_2^{\,4} - T_1^{\,4})$$

where σ is the Stefan-Boltzmann constant, ε is the effective surface emissivity, T_1 and T_2 are temperatures on surfaces 1 and 2. Note that T_1 and T_2 are values relative absolute zero temperature.

The new data format of the radiation model is:

$$\vdash \text{(mat)} — \text{RADI} — \sigma — \varepsilon — \text{(temp)} —$$

The convective and radiant heat transfer material models are only required for the FAT2 element which is used to represent convection and radiation.

Interfacing thermal and stress analyses

With the creation of the ASAS results database, the results from all major ASAS programs are now stored in a common standard format. Therefore, it has been possible to implement a facility to read in the temperature results directly from the database during the stress analysis run. This has a major advantage over the traditional approach in that it avoids the need to write out the full temperature load data into a text file as this can get very big in large models. Also, reading back the results only when required enables more flexibility in defining the load history variations.

The new data format for reading temperatures from the database is:

$$\vdash \text{GRES} — \text{sname} — \text{(lc)} — \text{(isub)} — \text{(node1)} — \text{(node2)} —$$

where

GRES	command keyword for reading data from the results database
sname	structure name containing temperature results
lc	user load case number in the URSD. By default, lc has the same user load case number as that specified in the stress analysis
isub	the load sub-case number (default 0)
node1	the lower limit of a node range where results will be transferred (default 0 means from lowest node on structure)
node2	the upper limit of a node range where results will be transferred (default 0 means to highest node on structure)

If specified, the GRES command must be given within the nodal temperature data block. For example, the following data will read the temperature results from case 10 of structure ABCD and apply them as temperature loads.

```
CASE 10 'Temperature load from thermal analysis'
TEMPERAT
GRES ABCD
END
```

Performing a transient heat run

A transient thermal job is invoked by specifying keyword HTRA in the JOB command (i.e. JOB HTRA). This apart, all data requirements are identical to those for the steady state heat jobs.

The heat capacity matrix C is analogous to the mass matrix in transient dynamics analysis. A generalised density data is required in the field (i.e. FIEL) material property data for calculating this matrix. The generalised density is the product of density and specific heat capacity (i.e. ρc). Any additional lumped heat capacity at a node can be specified in the direct input (DIRE) data in a way similar to the input of lumped added mass. For lumped heat capacity, the freedom code is T.

The time integration parameter γ_h controls the accuracy and stability of the solution. For linear problems, the solution is unconditionally stable if $\gamma_h \geq ½$,. The scheme with $\gamma_h = ½$ is called the Crank-Nicholson scheme or the trapezoidal rule. It is second order accurate but may suffer from wild oscillations. The choice $\gamma_h = 1$ is called the backward Euler scheme. It provides the necessary numerical damping to damp out the undesirable oscillations but is only first order accurate. The backward Euler is the only scheme that is unconditionally stable in non-linear problems. Due to the various advantages of the backward Euler scheme, the default setting for transient thermal analysis in ASAS(NL) is set to $\gamma_h = 1$. Other setting may be specified through parameter HGAMMA in the TEMP command (e.g. TEMP HGAMMA 0.5). The input HGAMMA value must be between 0.5 and 1.0 or else a warning will be flagged and the value of HGAMMA will be reset to the nearest value in the valid range.

In transient thermal analysis, the initial temperature distribution on the structure is specified in the INIT data block. All nodes without initial conditions defined (including those with prescribed temperatures) are assumed to have an initial temperature equals to the reference temperature *trefld*. If any non-zero initial condition (heat flux or temperature) is specified, a solution should be performed at the initial time in order to provide a consistent set of starting conditions for the time integration process.

The thermal results are saved using the RESU command. The table below shows the types of results saved with different keywords.

Result name	Keyword on RESU	Result type on URSD
Temperature	DISP	FIELD VARIABLE
Temperature rate	VELO	VELOCITY
Heat flux	STRS	FLUX
Heat field	STRN	FIELD

Units command will only affect input variables with temperature or length units. All heat associated properties (e.g. conductivity) do not have units conversion implemented and therefore they must be supplied in the global analysis units.

Convection and Radiation

The convection and radiation models described in Sec 2.2 are incorporated into the FAT2 element. An error will be reported if these material models are used with other element types.

For FAT2 elements with the convection or radiation model, the element length does not enter into the element formulation and thus the nodal positions of the element will not affect the calculations in any way. This is equivalent to having the natural stiffness option FLNS automatically turned on for those elements.

The temperatures employed in the radiation equation are relative to absolute zero degrees. By default, the temperature units are in degrees centigrade and the corresponding absolute zero value is set to -273.15. This value of absolute zero temperature is assumed by the program. If other units are used in the analysis, then the value of absolute zero temperature in the appropriate units must be defined using the PARA TAZERO command in order to model the effect of radiation correctly. For example, if temperatures are specified in degrees Fahrenheit, the command required is:

PARA TAZERO -459.67

Importing Temperatures for Thermal Stress Analysis

The GRES command is implemented within the nodal temperature data block as specified in Sec 2.3. It is assumed that the reading of temperature data through the GRES command is only allowed from the URSD (i.e. the 45 file) and so it is essential that this information is saved using RESU in the thermal analysis. In order to read from the 45 file, the thermal and stress runs must come from the same project. This means that the stress job must be an old job, i.e. JOB OLD, with the same project name defined as the thermal run.

The following are some comments regarding the implementation of the thermal-stress interface:

- As the 45 file is standard across the ASAS suite, the stress job can be ASAS or ASAS(NL).
- It is required that both the thermal and stress analysis jobs have the same node numbers for all structural corner nodes.
- The node range data (node1 and node2) enable certain nodes to be excluded from the stress analysis. This facility is useful if non-structural nodes have been used in the thermal model, for example, to model ambient temperature in convection or radiation boundaries.
- It is acceptable to have more than one GRES command in a load case.
- GRES can be specified together with ordinary nodal temperature data in the same load case.

Interfacing with FEMVIEW

The ASAS2FV program can transfer the thermal results to FEMVIEW. In FEMVIEW, temperatures (FEMVIEW attribute FIELVARI) and temperature rates (VELOCITY) are stored as nodal results while FLUX and FIEL are stored as element results.

The thermal analysis capability in ASAS(NL) provides a very comprehensive capability covering not only volume elements such as bricks but also shell and beam elements. ASAS can therefore tackle fire scenarios in buildings where construction uses different structural elements (plates, stiffeners, beams etc).

Job Replace

A new option JOB REPL (Job Replace) has been added to the header deck. Thus there are now three options NEW, OLD and REPL. The purpose of JOB REPL is to replace a previous run of the same structure or component within the same project. However, the replacement of a lower level component in a sub-structured analysis is not allowed in order to avoid the risk of accidentally deleting a job that will destroy the structure tree created. A file package error will be reported if such an attempt is made.

Benefit: Users no longer need to run COMPED to remove a reference to a model

ASAS2FV Enhancements

If beam code checks are selected from within the ASAS2FV panel a new load case LMAX is created to store the maximum unity check results across loadcases and transferred to FEMVIEW as element invariant results. In FEMVIEW when displaying the results using PRESENT NUMERIC, the unity check value is presented as a three part number in the form:

Loadcase number/result component integer/ maximum check values

The results from ASAS(NL) are now saved to FEMVIEW with analysis type set to 'load step'.

Benefit: This enables the plotting of results vs time graphs.

Coupled Wave Loading

With the coupled wave loading in ASAS(NL) a new option RVOF has been introduced which will switch off the relative velocity effect thus ignoring the structural velocities in the load calculation.

Benefit: In some cases the structures are quite stiff and this option allows the user to compare the effects of including or excluding the relative velocity effects. If the differences are small it means a simpler linear static solution may appropriate. (There may be other non-linear effects such as gaps, p-delta and soils).

In addition some codes of practice state that if the maximum deflection is less than the diameter of the tube then the relative velocity coupling should be ignored and only water particle velocities used.

Security Controls

The ASAS security control system has been revised. All existing users will be issued with updated security files.

There will also be a revision to the directory structure such that each version will have its own directory. This removes the risk of accidentally overwriting a previous version.

Surface Stress Consistency for Shell Elements

Prior to Version 14.02 there was a potential confusion between ASAS(L) and ASAS(NL) shell surface results and shell results displayed by FEMVIEW. Results from ASAS(L) or from ASAS(L) and ASAS(NL) via ASAS-POST were stored top to bottom in FEMVIEW i.e. surface 1 was the top surface. Results from ASAS(NL) and ASAS(NL) via POST-NL were stored bottom to top (i.e. surface 1 was the bottom surface). From ASAS 14.02 the latter method has now been adopted for all programs such that irrespective of the route to FEMVIEW ASAS modules use a bottom up format with surface 1 corresponding to the bottom surface in FEMVIEW.

The reason for adopting this is that ASAS(NL) uses through thickness integration points from surface 1 to n. It is particularly relevant for multi layered composite elements.

Note that the .ini file in FEMGV has been modified so that top and bottom surfaces are correctly recognised.

ASAS plate and shell stresses.

Prior to ASAS 14.02 it was necessary to run ASAS-POST in order to convert stress resultants to stresses. In ASAS 14.02 surfaces stresses can now be calculated in ASAS and results stored on the database. Von-Mises stresses at the top middle and bottom surfaces are also calculated and stored on the database.

Benefits: With the results stored on the ASAS Database (RESU option) the need for running ASAS-POST may be avoided and results transferred directly to the FEMGV database via ASAS2FV.

Also the stresses can be rotated to user required directions within FEMGV.

Hysteretic Damping

Hysteretic damping has been added for the SPR1/2 spring elements. This permits energy dissipation in dynamic analysis due to hysteretic material behaviour such as in non-linear soil-structure conditions e.g. earthquake analysis. The following gives details of its features and modelling definitions.

Inelastic Material Model for Spring Elements

The normal plasticity model forms the basis of the inelastic material model for spring elements. Figure 1 shows a typical skeleton curve that is used to represent such a model. The spring initially deforms in a linear elastic manner until the initial yield force fy1 is reached. Plastic yielding will start to develop when further loading is applied. If the loading is reduced or its direction reversed, the spring will unload elastically.

The unloading/reloading stiffness after yielding occurred may be different from the initial elastic stiffness. This is controlled by the material parameter α1. By default, the unloading stiffness is assumed to be the same as the initial stiffness (i.e. α1 is set to 1.0 if it is specified as 0.0).

The spring can have different response behaviour in the positive and negative loading directions. However, it is essential that the spring curve must pass through the origin.

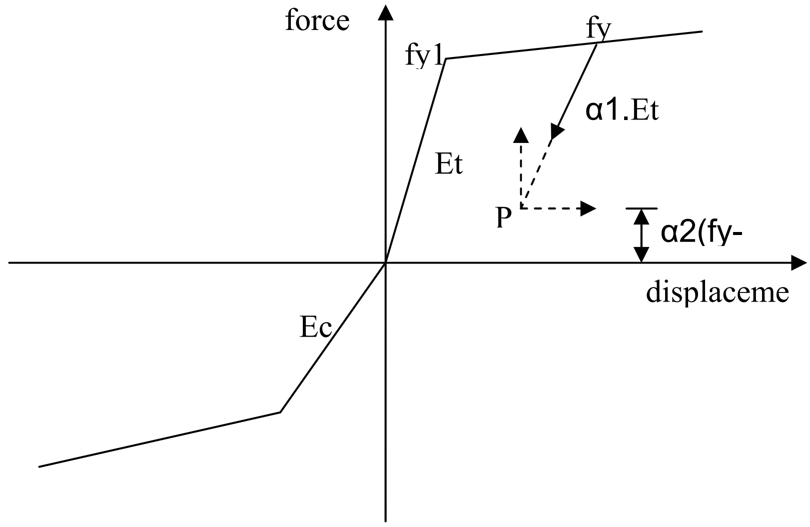


Figure 1 Skeleton Curve for Inelastic Spring

The hysteretic behaviour is modelled through shifting of the spring curve origin. Origin shift takes place when unloading continues beyond a limit point (point P in Figure 1). When this happens, P will become the new origin of the spring curve (see Figure 2). The material parameter α2 defines the location of point P and it allows for the modelling of different kinematic hardening behaviour. By default, α2 = 0.0 and thus the shifted origin is always along the horizontal axis. The setting α2 = 1.0 is almost equivalent to the pure kinematic hardening model in continuum elements. Normally, α2 should be set between 0.0 and 1.0.

As a result of the fact that the elastic stiffness may change after the first yield, the corresponding yield force fy1 may also be different from the initial value. Relative to the shifted origin, fy1 is determined from the intersection between the spring curve and the elastic stiffness line (see Figure 2).

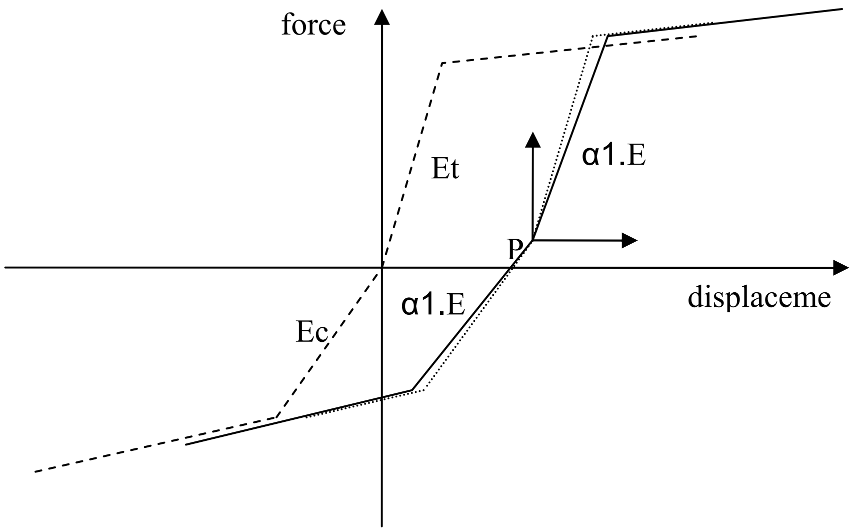


Figure 2 Spring Curve with Origin Shift

The hysteresis model can only be applied to model the spring behaviour. The dashpot behaviour will always remain as elastic.

Data Requirements for Inelastic Spring Analysis

In order that the same spring curve definition can be used in both elastic and inelastic analyses, the spring curve data are specified in the geometric property section as usual.

The inelastic behaviour is activated through keyword PLAS in the TITL/PROB or GROU command. With PLAS specified, plastic material properties must also be supplied. It is proposed that the data syntax of the inelastic spring model is as follows:

|— (mat) —— PLAS —— SPRG —— α1 —— α2 ——

where

mat	material property integer
PLAS	keyword to define the material as plastic
SPRG	inelastic spring model keyword
α1	unloading elastic stiffness factor (set to 1.0 if specified as 0.0)
α2	kinematic hardening factor

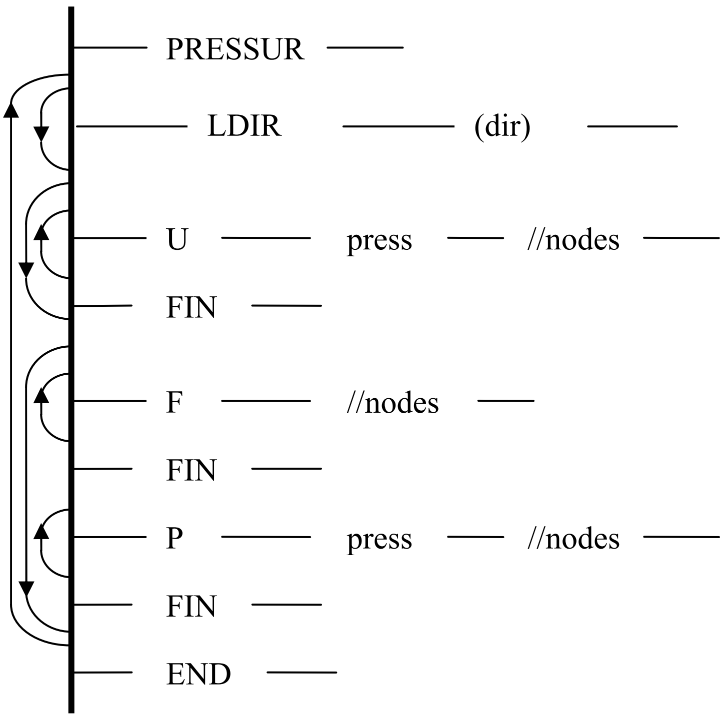
Extension of User Material Interface to permit user defined models

The user material facility is extended to include spring elements in order to allow the user to develop his/her own hysteresis/inelastic model. When using the user material interface, all spring data must be defined through material properties and not geometric properties. In this case, all spring geometric properties will be ignored and only dashpot geometric properties will take effect. If no dashpot is present, however, dummy spring geometric properties must be provided to avoid program error.

Directional Pressure Loading

Prior to ASAS 14.02 it was only possible to apply pressure loads normal to a surface or line. With ASAS 14.02 it is possible to apply pressure loads in global directions and in the case of shell elements in element local axis systems also.

The revised pressure load data format is modified as specified below to allow for directional pressure loading.



As previously, U is a keyword to define uniform pressure, F is a keyword to define data as face data definition and P is a keyword to define data as nodal pressure values.

LDIR is the new command for specifying the loading direction of pressure. The command is optional and normal pressure is assumed if this is absent. This ensures compatibility with existing data format. If LDIR is present, the loading in the subsequent pressure data (i.e. U or F/P commands) will act in the specified direction until the next LDIR command is encountered. The LDIR command must be specified immediately following the PRESSURE load type header or the FIN command after a set of U or P data.

The direction of loading is specified through parameter *dir*. The setting of *dir* can be one of the following:

dir	=	GX	global X direction
		GY	global Y direction
		GZ	global Z direction
		X	local X direction
		Y	local Y direction
		Z	local Z direction

If *dir* is left blank then normal pressure (i.e. default direction) is assumed.

The directional pressure loading facility is implemented in all programs that accept pressure load. This includes ASAS, RESPONSE and ASAS(NL).

Benefits: loads acting in the direction of gravity can be applied. Also traction loads as may occur in transportation can also be applied.

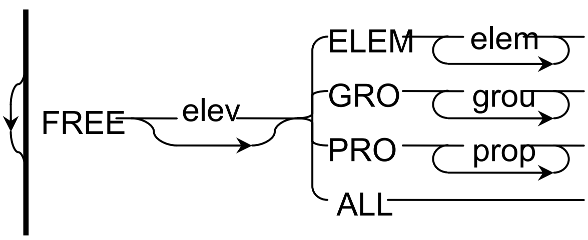
Modified Methodology for ASAS-WAVE pressure calculations

This development covers two parts:

Internal Pressure in Flooded Members

The free flooding option in WAVE allows the user to specify a list of elements which are flooded. Prior to ASAS 14.02, the free flooding information (FREE command) was used only for calculation of the weight of the member. The BUOY command prompts calculation of external endcap forces based on the static water pressure to the wave surface. The BUOY command allows specification of the density and proportion of flooding of the member, but uses this information only for calculation of the self-weight of the member. Under this method the compressive forces in a flooded member can be excessively large due to the external endcap forces being present but not the internal end-cap forces.

In ASAS 14.02 the users can specify the water level for calculation of the internal pressure in the element. Users can specify the water pressure to be taken to the wave surface, or the still water surface or possibly to another level. Thus the FREE command is modified as specified below to allow the required flexibility in specification of the data. This format also allows existing data to run with no changes to data or results.



The new piece of information on the FREE data line is the elev data. If elev is left blank then the program calculates as at present – i.e. no internal hydrostatic end-cap forces. If elev is set to a value that is higher than the seabed elevation then internal end-cap forces are calculated assuming water pressure to elevation elev. For points that are above the specified elevation elev, the internal end-cap forces are zero. If elev is set to be less than the seabed elevation then internal end-cap forces are calculated to the elevation of the wave surface vertically above the node in question.

Note that dynamic pressure concepts discussed below are not applicable to internal end-cap pressures.

Improvements to method of calculating hydrostatic pressure

Rationale: - Hydrostatic pressure is used in the calculation of end-cap forces (rigorous buoyancy only). Prior to ASAS 14.02 the method used for the calculation of the pressure to be used in this calculation is to take $\rho * g * zws$ where zws is the distance from the point to the wave surface.

Consequently in ASAS 14.02 a new method has been introduced based on the Navier-Stokes equation $-\frac{\partial p}{\partial z} = \rho \frac{dw}{dt}$ (note: acceleration on the r.h.s. is total). This equation can be used assuming a boundary condition of zero pressure at the water surface, then integrating the acceleration profile through the depth to give the pressure profile.

Advantages of this method over other methods are as follows:

a. It is completely general and the same code should be valid for all wave theories.

b. Pressures at the water surface will be zero; the errors are towards the bottom where they will have less effect on over turning moment calculations.

c. In shallow water the pressure gradient at the sea-bed will be correct.

Benefit: For the design of new jacket structures the new method can save considerable weight, particularly for large wide jackets. For existing structures which are subject to structural modification as part of an asset management programme the new methods can avoid major structural changes and thus save considerable costs.

ASAS-VISUALIZER

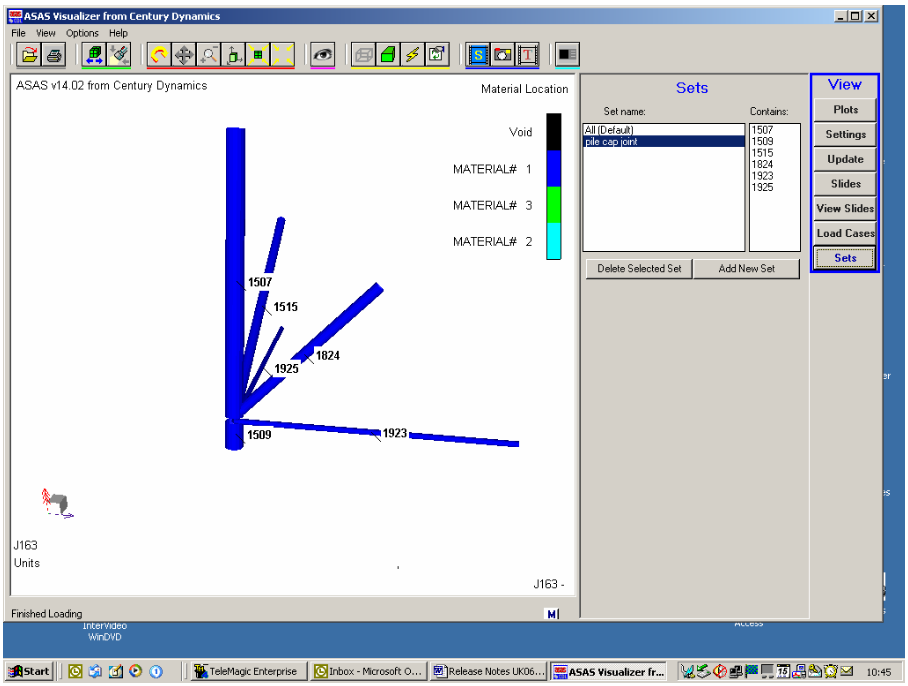
Since the release last year of the ASAS-VISUALIZER a range of new capabilities has been added in as defined in Phase II but also in response to user feedback.

Viewing model and results by SETS

In ASAS 14.02 the use of sets can be employed using 3 different systems.

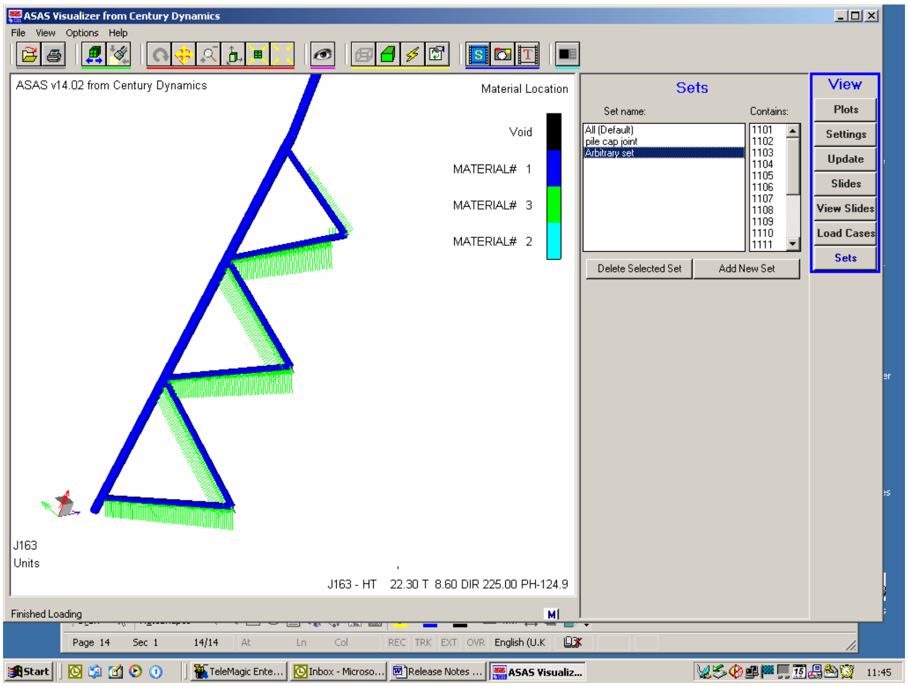
Building up a selection by selecting the set widget, clicking on interactive, selecting the ‘Examine’ button and then using the LH mouse click to identify those members you wish to be included in the set. Note that the ‘Use Sets’ box needs to be ticked.

This shows a selection of members selected by the mouse to constitute a pile cap joint. Results such as unity check values can also be displayed on the display.



1) Choosing sets by Mouse

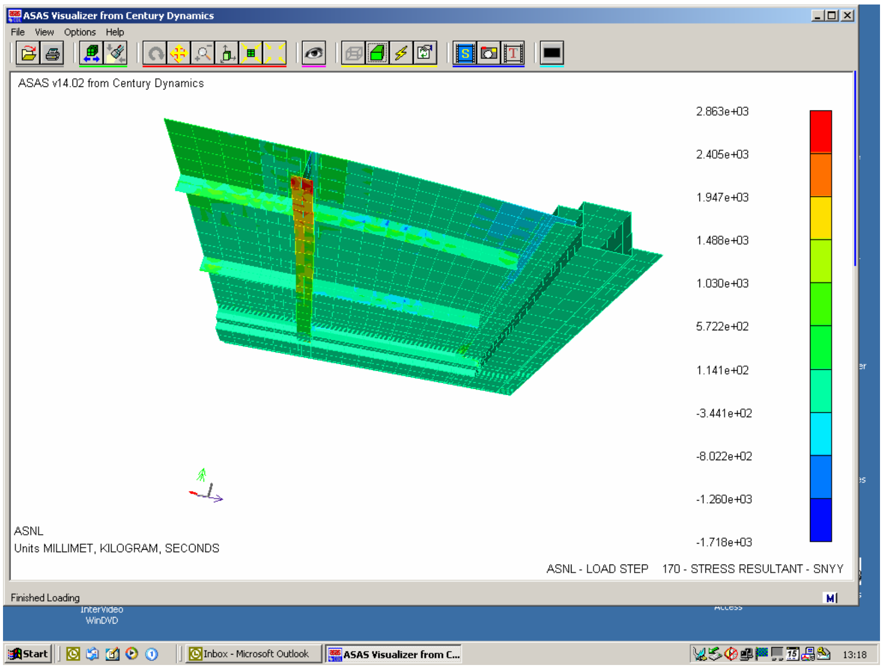
The next example shows sets selected from the list of elements. In this case members connected to a leg of a jacket. Note also that distributed forces resulting from wave loading are also shown (in green).



2) Choosing SETS from list of elements

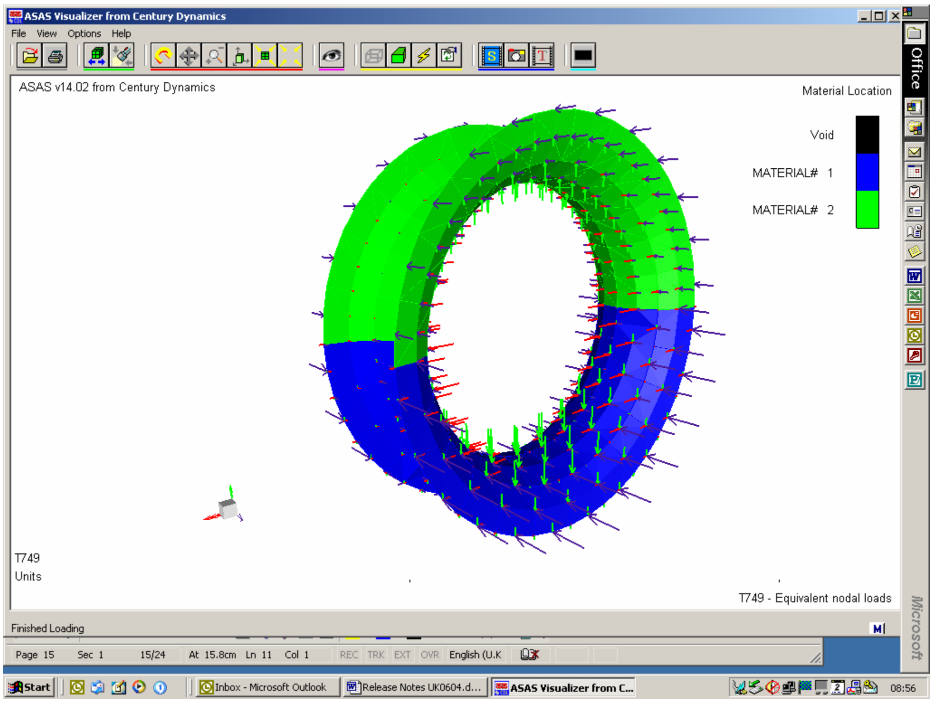
CENTURY DYNAMICS

The image below consists of a set which has been predefined within the ASAS data deck. In this case contours of stresses are shown for a particular load step.



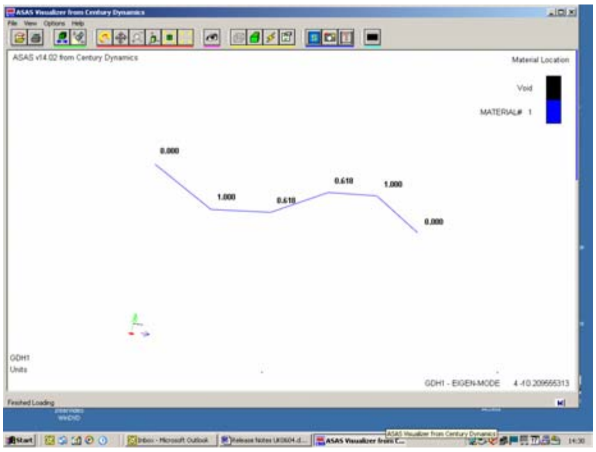
3) Selecting a pre-defined ASAS set

Applied Load Values displayed by vectors



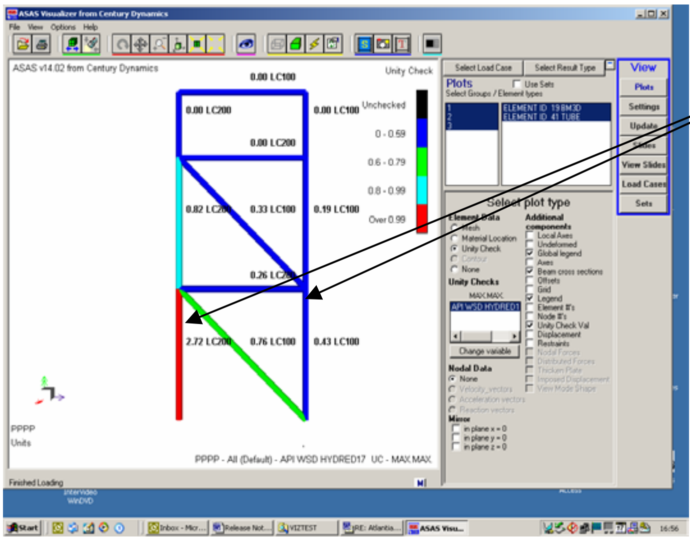
CENTURY DYNAMICS

Automatic Mode Shape Animation



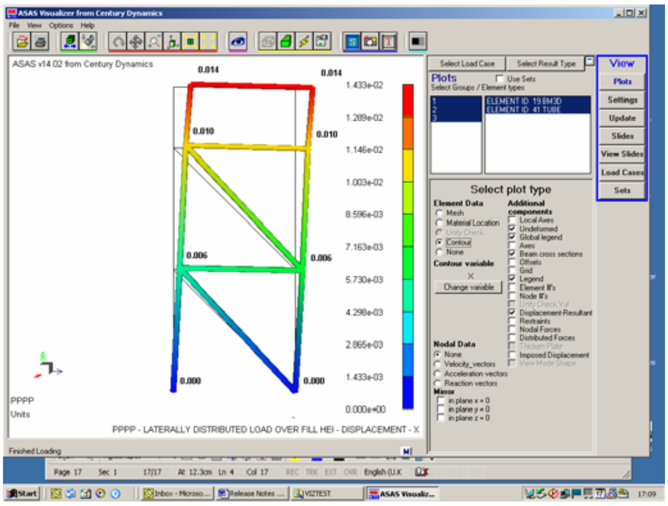
The ASAS-VISUALIZER can show animations of mode shapes with or without beams as section. The animation can be stopped at any time.

Extract and display the maximum value across load cases

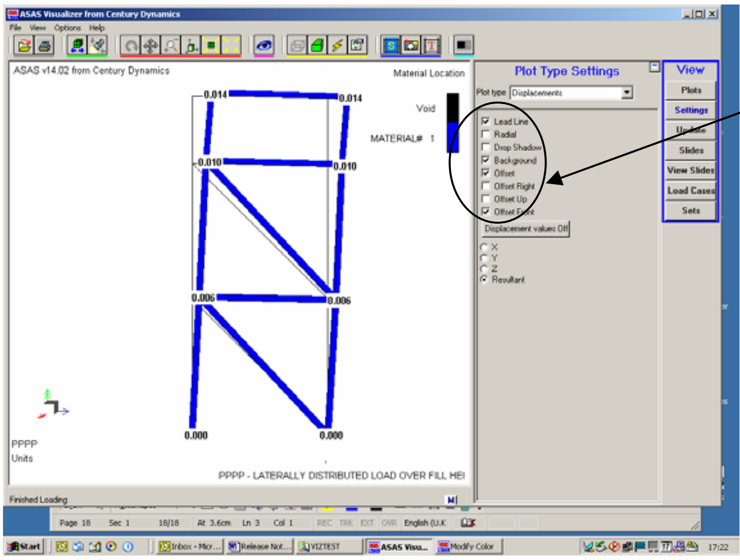


By selecting all load cases for particular variables the ASAS-VISUALIZER identifies the actual value and corresponding load case. In this example they are max-max unity checks.

Displacements can be displayed as numeric, contour and deflected shape.

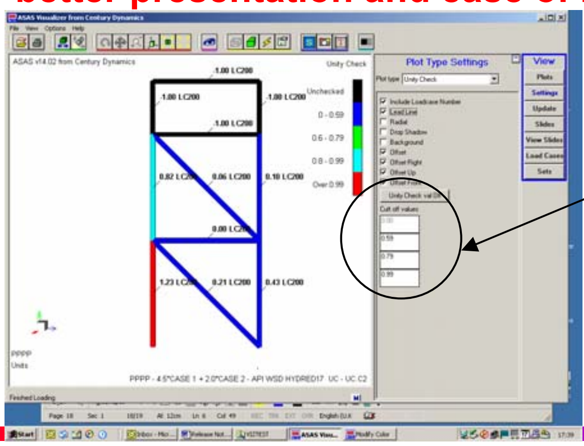


Users can define font sizes and display style for clarity



Note the range of options to ensure you get the best visible display of characters.

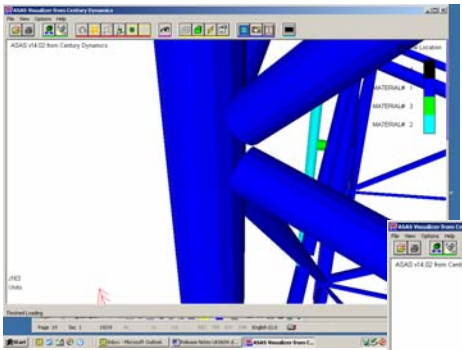
Users can vary the bands which define unity check values for better presentation and ease of results checking



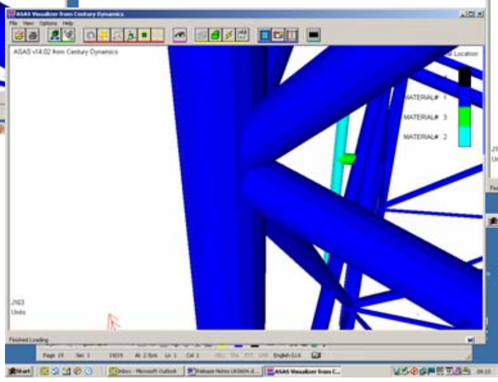
Users can set the bands to suit requirements to provide the best display

Beam offsets can now be displayed

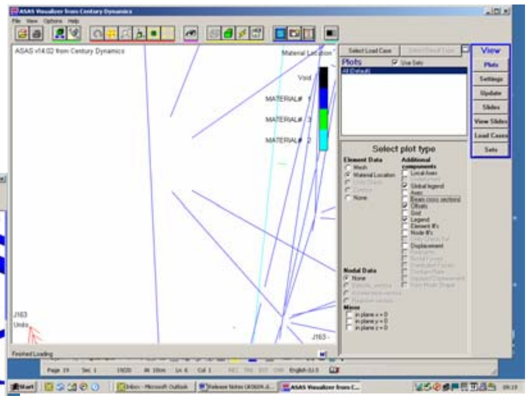
Offsets with beams sections displayed



Tube sections with offsets not displayed



Same joint showing offsets with elements represented as line elements



Phase II of the ASAS-VISUALIZER as part of ASAS 14.02 marks the point after which ASDIS, PICASO and BEAMVIEW will no longer be provided as part of the installation. The ASAS-VISUALISER provides many capabilities which are superior to these older products which use a different compiler which is no longer compatible with later versions of Microsoft operating systems. For users of BEAMVIEW it will be noted that the ASAS-VISUALIZER does not provide a direct feature by feature replacement. However overall users should find requirements can be met. The flexibility provided by sets, groups and materials in particular will be extremely useful for graphical report creation.

FEMGV

FEMGV 7 will be released shortly and will be the subject of separate release notes. In the meantime we have made enhancements to the interface program which enables more features within ASAS to be defined within the FEMGV system.

AQWA 5.5

Installation Changes

With AQWA 5.5 users can now specify which drive they would like the software to be loaded onto. Previously this was drive 'C'. In addition users will have the option of retaining previous versions of AQWA alongside the latest release. This is sometimes necessary for large projects when the same version number is required to be used throughout usually for QA purposes.

Security Controls

The AQWA security control system has been revised. All existing users will be issued with updated security files.

There will also be a revision to the directory structure such that each version will have its own directory. This removes the risk of accidentally overwriting a previous version.

New developments

The last version that was released as a full installation was version 5.4E. For completeness the major developments since then are summarised below. Details of all developments and bug fixes can be found in the PSR lists on our website; www.centdyn.co.uk.

Extension of Convolution Memory

Until version 5.4F the convolution method used an impulse response function with a duration of 60s. In a few cases it was found that this was not long enough to allow the response to decay sufficiently, and this could lead to significant inaccuracy in subsequent calculations of added mass and radiation damping. The duration of the impulse response function has been increased to 120s.

Increased Solution Speed

Changes have been made to increase the speed of AQWA-LINE when solving for the source strengths. This reduces the time to perform the matrix solution by about 50%, but it does not have any effect on other parts of the analysis.

Import of QTF database

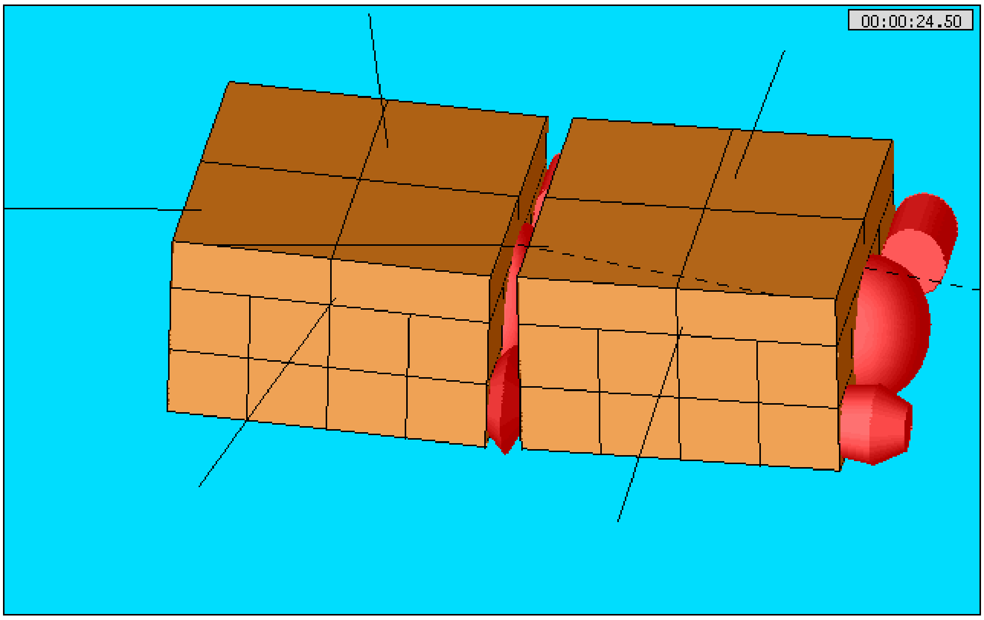
A full set of QTF sum and difference frequency coefficients can now be imported into the AQWA database via the .dat file. In addition, the facility for the import of added mass terms has been extended to include coupled terms for interacting structures.

It is now possible to import all the hydrodynamic coefficients, overwriting those calculated by AQWA-LINE, thus allowing the user to define the hydrodynamic database with data from model tests or from another program.

Note that non-hydrodynamics parameters, such as interaction distances, will still be calculated by AQWA-LINE.

Fender Moorings and constraints

The current suggestion for modelling fenders in AQWA is to use a “trick” with mooring lines. Now a new type of mooring has been added to model a fender more realistically. There are three types of fender moorings: fixed non-directional, fixed directional and floating. They are input in Deck 14 with the rest of the mooring data, and are included in the total number of moorings, which cannot exceed 100. The picture below illustrates the three types of fender in a test problem with two Faltinsen boxes.



User- defined forces via Dynamic Link Library (DLL)

For many years AQWA has been able to accept external forces defined in an .XFT file. However, these had to be defined before the analysis was carried out. It is now possible for the user to create his or her own .dll to calculate a force based on the time, the position and/or the velocity of a structure. A mass matrix can also be defined at each time-step to simulate inertia forces. The calculation can be controlled by a set of up to 100 integer parameters and 100 real parameters that may be input in deck 10 and will be passed to the external force routine.

Examples of FORTRAN and C routines to create the DLL are provided in the documentation.

This facility could be used to model, for example:

- a dynamic positioning system
- a steering system.
- the towing force provided by a tug
- a damping system with unusual characteristics
- the suction force between two ships close together, or between a ship and the sea-bed.

Friction in Articulations

In addition to stiffness and linear damping it is now possible to model Coulomb friction in articulations. This is done by specifying four coefficients on a new FCON card in Deck 12. Three of the four coefficients are similar to traditional friction coefficients from which the friction is calculated as a proportion of the reaction. They relate to the friction about the axis of rotation to:

1. the transverse force
2. the overturning moment
3. the axial force

The fourth coefficient gives a constant moment.

Usability improvements

Following feedback from users a large number of small modifications have been made which are intended to make the program easier to use or understand. Several simply involve clearer terminology in error messages or output and will be useful to new users. The other modifications are summarised below.

• AQWA-Graphical Supervisor modifications

1. The viewpoint can now be changed without limit.
2. Elements can be selected using a drag box. The selected elements may then be displayed or omitted from the plot.
3. A very simple representation of the sea-bed can be plotted
4. It is possible to scale the graphic used for in-line buoys, which can sometimes be difficult to see when the vessel is large.
5. Graphs can be output direct to a printer.
6. The shear force and bending moment calculations can now be carried out using only the vertical component of hydrostatic pressure. This enables a direct comparison to be made with conventional SWBM plots.
7. Graphs can be plotted against period instead of frequency.
8. A list of the most recently opened files is provided.
9. The default moments of inertia for models generated using the AGS have been changed to reflect values conventionally used in the industry.
10. A new status bar has been added which gives help for many of the AGS calculation functions when the cursor is placed over a menu entry.

- Modifications to batch programs

1. The programs now check for coincident nodes on an element.
2. The MSTR card now allows a yaw angle to be input. NOTE; use of this can lead to confusing values for RAOs and similar results.
3. AQWA-NAUT can now read an .eqp file using the RDEP option.
4. In many places path-names have been extended to 255 characters.
5. AQWA-LIBRIUM now reports the angle of mooring lines at fairleads.
6. Backing files to be copied can be specified on the restart card. This removes the need to copy .res, .eqp and .pos files from one run to the next.
7. There is a new CLMP card in Deck 14 for specifying clump weights.
8. AQWA-LIBRIUM now produces a .plt file. This allows progress towards equilibrium to be plotted in the AGS, which can be very helpful when trying to understand the behaviour of a model that does not converge.

Modified AQWA-TETHER Calculations

AQWA has had the capability to model tethers for some time. However, it was found that the calculations could become less accurate when modelling some of today's installations, which may be in very deep water. The calculation method and stiffness matrix used for tether elements have been improved to give more accurate results, particularly where long elements and top restraints are used.

BUG FIXES

Fixes to Grade 4 and 5 bugs are summarised below. These are bugs that produce incorrect answers: Grade 4 bugs give obviously incorrect results while Grade 5 bugs give errors that are not obvious to the user. Details of all the bug fixes can be found on our web-site.

Grade 4/5 Bug Fixes in 5.4F – 5.4H

There were no Grade 4 or 5 bug fixes in versions 5.4F to 5.4H

Grade 4/5 Bug Fixes in 5.4J

The mirror node generation facility used to ignore the STRC card.

Grade 4/5 Bug Fixes in 5.4K – 5.4M

There were no Grade 4 or 5 bug fixes in versions 5.4K – 5.4M

Contacts for further information:

All countries except Americas and Nordic Countries:	Americas	Nordic Countries
Phil Cheetham Century Dynamics Dynamics House Hurst Road Horsham, West Sussex RH12 2DT T +44(0)1403 270066 F +44(0)1403 270099 E all@centdyn.demon.uk	Paul Schofield Century Dynamics 11200 Richmond Avenue Suite 300 Houston TX 77082 T +1 832 295 4445 F +1 281 496 1225 E CDHouston@centdyn.com	Kai Viggo Munch Teknisk Data as P.O.Box 6655 Etterstad N - 0607 Oslo Norway T +47 22 660980 F +47 24 071519 E asas@tda.as