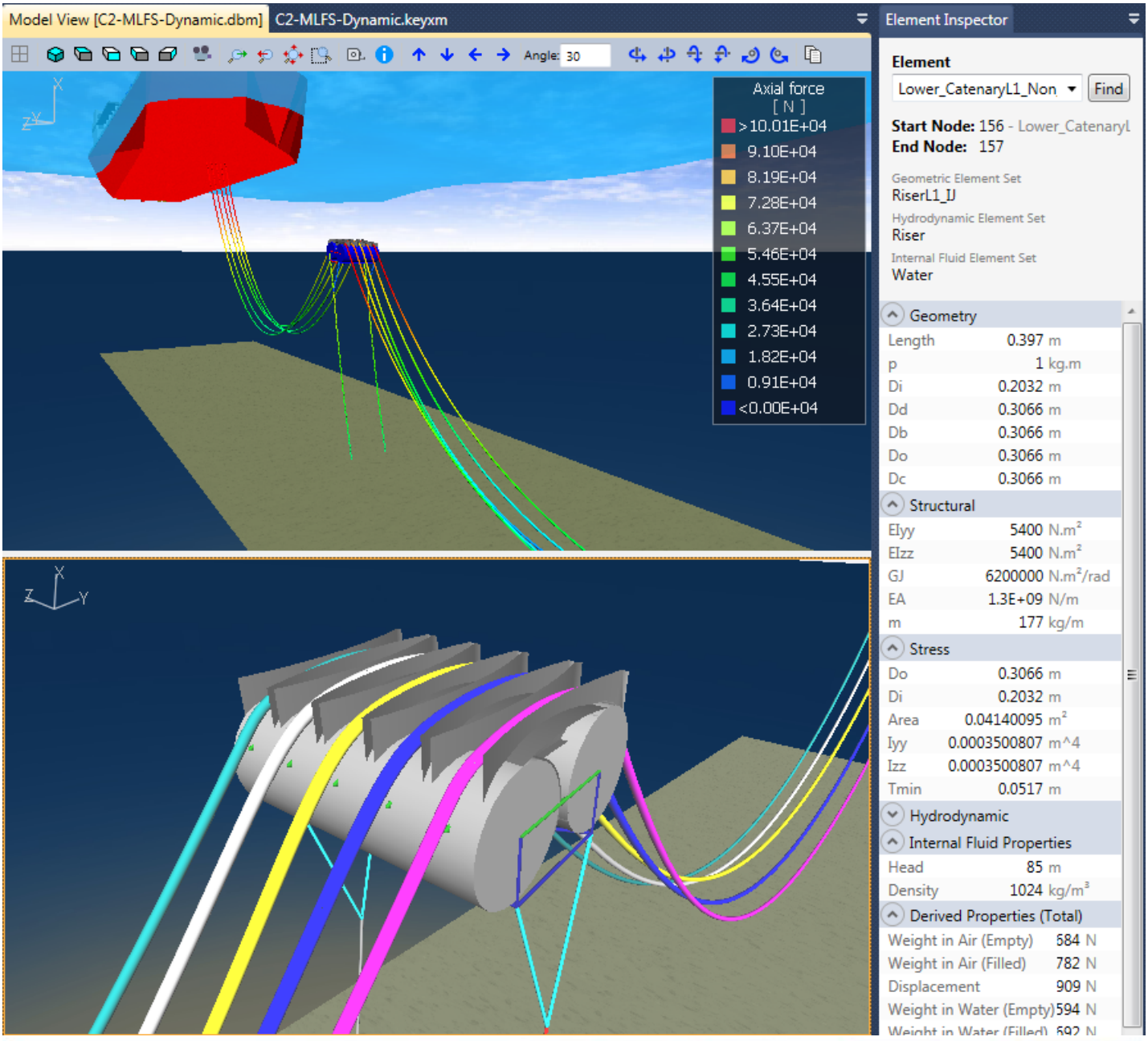
Flexcom NEWS UPDATE

August 2016



WELCOME

Aengus Connolly
Flexcom Product Manager

I am pleased to announce the release of Flexcom 8.6. The overarching theme of this version is enhanced user experience. Guided by feedback from our global user base, we are proud to deliver quite a number of very practical advances in key areas.

While the focus of this release is primarily on improved productivity and enhanced user experience, we continue to work on a number of significant technological advancements which we look forward to making available to you in the near future. User feedback is an essential part of this process. We invite you to join the conversation and welcome your opinions and suggestions which will in turn help guide future development. Feel free to call me at the number below, or simply email any suggestions to software.support@woodgroup.com.

In spite of the difficult conditions our industry finds itself in, Wood Group remains committed to product development to ensure that we continue to provide tangible value to our customers.

We hope you enjoy Flexcom 8.6.

Best regards,
Aengus.

Aengus Connolly
Flexcom Product Manager
Wood Group
T: +353 (0) 91 48 1238 (Direct)
E: Aengus.Connolly@woodgroup.com

Follow Flexcom on LinkedIn for the latest news and updates…

Flexcom 8.6 Highlights

The latest version provides several important new features along with a range of practical additions which contribute to improved user experience. Highlights include…

User Experience: The introduction of the Flexcom 8 series provided a step change in terms of software usability. Flexcom 8.6 continues that trend and we are confident that it is our most user-friendly version yet.

3D Plotting: 3D Plotting is an advanced feature which provides enhanced visualisation of engineering data. For example, you can plot maximum effective tension as a function of both wave period and incident wave heading in a 3-dimensional space.

Model Building: A new feature has been added which allows you to model repeated line sections quickly and easily.

User Solver Variables: You can directly augment the global force vector to simulate an arbitrary time-varying load. It is even possible to directly modify the constitutive finite element matrices should you have very specialised modelling requirements.

Run-Time Performance: Flexcom 8.6 has shown to be approximately 5% faster on average than Flexcom 8.4 for time domain simulations.

Pipe-in-Pipe Models: It's now possible to alternate between fixed and sliding connections, affording you greater modelling options.

Damper Element: Velocity-dependent damper elements allow you to simulate more advanced control systems.

Additional Outputs: Additional outputs provide greater transparency regarding internal computations which are carried out at run time.

Fault Corrections: Flexcom 8.6.1 corrects a number of program faults identified in the preceding version, Flexcom 8.4.4

Software Download

The Flexcom 8.6 installation pack is available for download from our website.

Download Flexcom

To install the upgrade, save the ZIP file to a temporary folder on your hard drive, unzip the contents, run 'InstallFlexcom.exe' to launch the Setup Wizard, and then simply follow the on-screen instructions.

Contents

User Experience

The introduction of the Flexcom 8 series provided a step change in terms of software usability and advanced user experience. Flexcom 8.6 continues that trend and we are confident that it is our most user-friendly version yet, offering a host of productivity enhancing improvements.

By way of background, some of the more complex models examined by our technical support service contained a couple of thousand lines of keyword data. Feedback from our global user base indicated that the keyword editor struggled to cope with such large volumes of input data.

We have utilised advanced profiling tools to examine the user interface source code, identified potential computational bottleneck areas, diagnosed the root causes, and implemented several key optimisations. This development work has culminated with the delivery of a highly efficient Keyword Editor. The editor is now highly responsive, with no time lag between user command and software response. Users can still avail of all the helpful features such as data prompting and auto-completion, but now with speed and responsiveness comparable to a generic text editor.

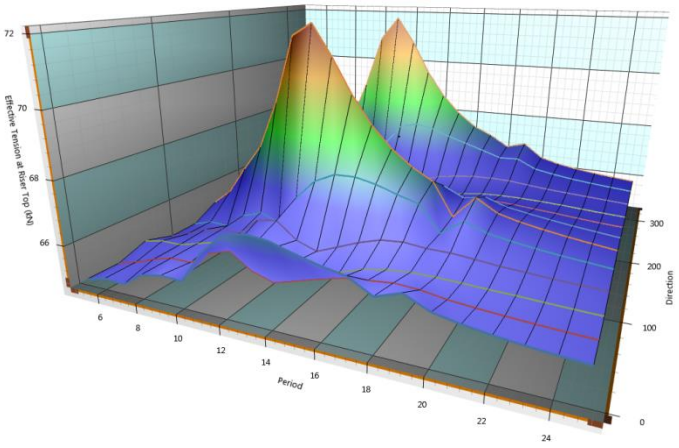
```
*ADDED MASS
  FLOATING BODY=BUOY1
  FILE=E2-CBC-Added Mass.incx

*RADIATION DAMPING
  FLOATING BODY=BUOY1
  FILE=E2-CBC-Radiation Damping.incx

*FORCE RAO
  FLOATING BODY=BUOY1
  FILE=E2-CBC-Force RAO.incx
```

Additionally, several keywords now have the ability to reference data from external files, via a FILE= keyword entry. This is particularly useful for coupled analysis simulations, where large amounts of input data are required to define added mass, radiation damping, force RAO data etc. Referencing external files removes clutter from the main keyword deck, and also reduces keyword processing workload thereby further improving user experience. Learn More >

3D Plotting

3D Plotting is an advanced feature which provides enhanced visualisation of engineering data. One (dependent) variable is plotted against two other (independent) variables in a 3-dimensional space. The plot control is fully interactive, and you can pan, zoom and rotate the viewpoint for ease of inspection. Helpful tooltips allow you to retrieve exact data and any point in the 3D space. It's also possible to switch back to planar/2D views should you prefer.

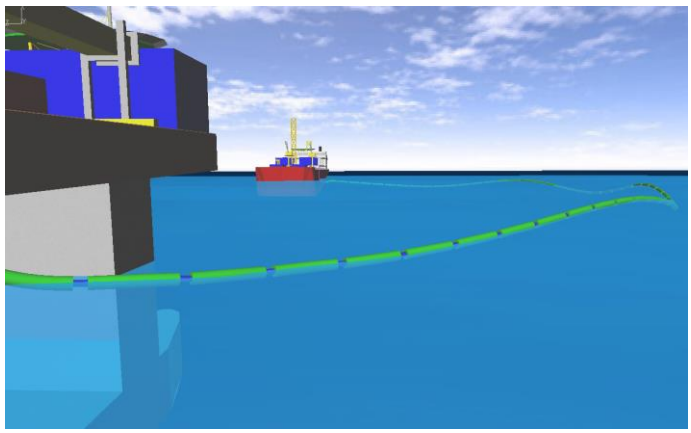
3D plots are ideally suited to the examination of vessel RAO data, but more importantly to summary postprocessing and collation. If you are performing a series of analyses (for example to examine a large number of different load cases), the summary collation facility provides a useful means of assembling all the pertinent summary data across a range of load cases into a single summary collation spreadsheet. Enhanced data visualisation is now provided by the ability to produce 3-dimensional summary collation plots. Here you can plot the variation of any summary postprocessing output against any key driving parameters. For example, you can plot maximum effective tension as a function of both wave period and incident wave heading in a 3-dimensional space. Learn More >



Model Building

Modelling Repeating Sections

A new feature has been added which allows you to model repeated line sections quickly and easily. A group of sub-sections may be defined under a *LINE SECTION GROUPS keyword, which may then be subsequently referenced under the normal *LINES keyword as required. When the repeat section group option is invoked, Flexcom automatically populates each line section with as many of the sub-section groups as will fit between the specified start and end distance for that line section. If the repeated group length doesn't divide into the distance evenly, you will be notified to that effect, and the last section group truncated appropriately.



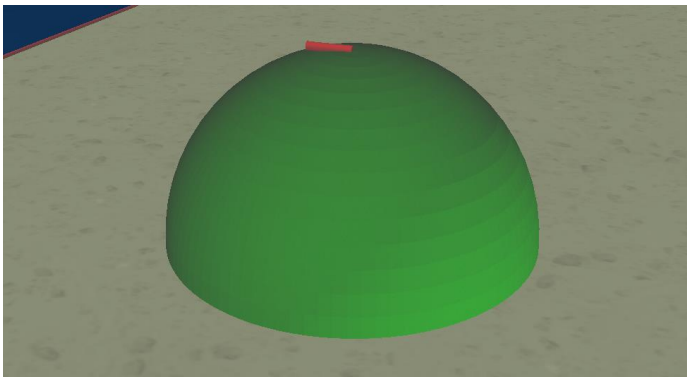
Consider for example the sample floating hose model which is shipped with Flexcom. It contains 52 identical sections of buoyed hose, separated by standard/unbuoyed hose sections in between. This geometry required over 230 lines of keyword data to construct using Flexcom 8.4.4. The new functionality in Flexcom 8.6.1 results in an 8% reduction in volume of required keyword data. Learn More >

User Solver Variables

Flexcom has traditionally provided a number of basic user subroutine options to accommodate specialised loading such as unusual current profiles (e.g. solitons), arbitrary loading (e.g. load time histories) and boundary conditions (e.g. load time

histories). These options were somewhat limited in the sense that the arbitrary loading was largely prescribed beforehand, with no dependence on instantaneous solution parameters which evolve as the simulation progresses.

Flexcom 8.6 provides advanced user subroutines, denoted as 'user solver variables' to distinguish from legacy features, which provides much greater flexibility and control. The user is empowered by the ability to access a host of solution variables including nodal kinematic variables (displacements, velocities, accelerations) and elemental restoring forces (effective tension, bending moment, curvature, torque etc.). Equipped with this information it is then possible to directly augment the global force vector to simulate an arbitrary time-varying load. It is even possible to directly modify the constitutive finite element matrices (i.e. stiffness, mass). Naturally this requires some programming expertise, but it provides complete generality for power users. Learn More >



Some illustrative examples are provided with Flexcom 8.6 to help upskill users and demonstrate the potential of this powerful new feature. J04a - Follower Force illustrates the application of arbitrary loading. The direction of the applied force is dependent on the instantaneous position of the point of application. The use of custom code enables the definition of a force which can track/follow the direction of response, via continuous adjustment of the global force vector. J04b - Sphere Contact illustrates how to model contact with a spherical surface, a modelling feature which is not currently available as standard in Flexcom. This is a more complicated example which involves augmentation of both the global stiffness matrix and the global force vector. Learn More >

Run-Time Performance

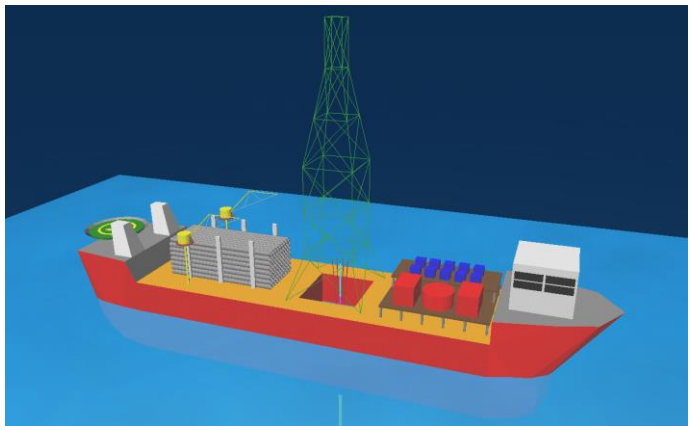
You may notice an improvement in run-time performance after upgrading to Flexcom 8.6. We have incorporated some source code optimisations which have resulted in increased computational efficiency. Based on benchmark tests across a range of sample analyses, Flexcom 8.6 has shown to be approximately 5% faster on average than Flexcom 8.4 for time domain simulations.

Run-time benchmarks were performed on a standard 64-bit machine with a Windows 7 operating system. The test cases include a selection of the standard examples which accompany the software, plus some real-world models received from external clients. Note however that any performance increases are application specific (i.e. depending on the model size, loading regime etc.), and will also be affected by external factors such as operating system, hard drive access etc.

Pipe-in-Pipe Models

Sliding Connections

Sliding Connections are appropriate for modelling scenarios where there is significant relative axial motion between the primary and secondary structures. In certain circumstances, it may be desirable to allow the software to initially determine appropriate connections between a primary and secondary section, and to subsequently treat these connections as standard connections. For example, when setting up a landing string model, the initial set-down of the drill string within the marine riser can make it very difficult to manually identify the optimal set of pipe-in-pipe connections in advance of the initial static analysis. Designating the connections as sliding allows the program to automatically determine the optimal connections, minimising effort on the part of the user. While there may be significant relative axial motion between the initial model definition and the converged static solution, there is comparatively little axial motion during the actual simulation itself (e.g. when the model is subjected to wave loading). Invoking a new keyword (*NO PIP SLIDING) keyword in subsequent restart analyses thereby ensures computational efficiency (any overhead associated with the monitoring of nodal locations is eliminated), and can also provide enhanced numerical stability (connectivity of the finite element model remains consistent throughout the simulation). Learn More >



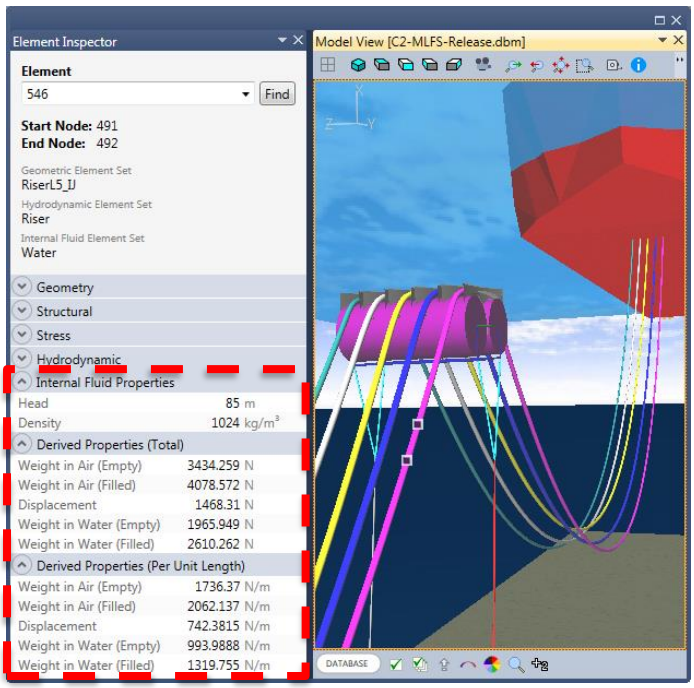
Documentation

Documentation regarding pipe-in-pipe theory has been strengthened in order to provide greater transparency to the Flexcom user regarding the internal workings of the software. In particular, the theory extensions include comprehensive sections on the simulation of *Contact Modelling* and *Hydrodynamic Forces*. Fundamentally, there is more information available to the user. The overall word count for pipe-in-pipe theory was about 3,160 in Flexcom 8.4. The new documentation for the same sections now stands at 8,440 words, an increase of almost 170%. Learn More >

Damper Element

The resistance of damper elements can now be controlled as a function of velocity. This gives you much more control over the damper characteristics and allows you to simulate more advanced control systems. Given that damper elements are sometimes used to simulate power extraction, Flexcom also provides additional post-processing options in this area. Learn More >

```
*DAMPER
 {DamperElement}, C1=VelocityDependence

*DAMPER DATA
 NAME=VelocityDependence, VARIABLE=VELOCITY
 0.0, 0
 0.5, 1000
 1.0, 3000
```



Additional Outputs

The range of additional outputs available for user inspection has been extended in Flexcom 8.6, providing greater transparency to the Flexcom user regarding internal computations which are carried out at run time.

```
*PRINT
 OUTPUT=
```

AUXILIARY DATA
CENTRIFUGAL FORCES
CONVECTED ELEMENT AXES
CRITERIA
DETAILED ELEMENT OUTPUT
FREQUENCY DOMAIN SOLUTION
PIP CONNECTIONS
REYNOLDS COF OUTPUT
SLUG FLOW
WATER PARTICLE HYDRODYNAMICS
WAVE DISCRETISATION

Any of the following outputs may now be requested via the *PRINT keyword.

- For models which contain sliding pipe-in-pipe connections, it is now possible to monitor the connections over time, and to identify with certainty which nodes are connected together at every solution time step.
- The new *Water Particle Hydrodynamics* option provides detailed output regarding the spatial and temporal distributions of water particle velocity and acceleration. It is possible to inspect these terms for on an element-by-element basis, in order to ascertain the applied loading at any point in the wave field. Learn More >

The *Element Inspector* has also been extended to present derived properties, such as wet and dry weight, total displacement, including or excluding internal fluid. Learn More >

Fault Corrections

Flexcom 8.6.1 corrects a number of program faults identified in the preceding version, Flexcom 8.4.4. The fault corrections are as follows:

Issue 1: Motion Database File

- Location: Database Files
- Severity: Trivial
- Description: The *DATABASE CONTENT keyword may be used to customise the contents of the motion (DBM) and force (DBF) database output files, typically to conserve disk space. However the motion database file is always created, even if the *Motion*, *Velocity* and *Acceleration* outputs are all suppressed.

  Further background information is as follows. When custom geometric properties are specified under *PROPERTIES, results such as axial force, von Mises stress and hoop stress need to be back-calculated from the effective tension and pressure. Refer to Axial Force and Effective Tension for further details on this computation. Nodal motion/position was required to calculate internal and external hydrostatic pressure and so was enabled, despite any options specified under *DATABASE CONTENT.
- Workaround in Flexcom 8.4.4: Unwanted DBM files may be manually deleted post-simulation to save disk space.

Issue 2: Running Fatigue Analysis Based on Timetrace (GRD) Outputs

- Location: Time Domain Fatigue Analysis
- Severity: Minor
- Description: Flexcom requires access to time histories of axial force, Y bending moment and Z bending moment at each location (hot spot) of interest. This information can be stored via Timetrace Output (traditional approach) or Database Output (more popular, and recommended, option in recent times). If you have chosen to store results via timetrace output only (i.e. GRD files), Flexcom searches for a restart file (RST), even though this is not necessary, and terminates the analysis if it cannot find one.
- Workaround in Flexcom 8.4.4: Manually create the required restart files if they are not present.

Issue 3: Scale Factor for Stresses

- Location: Time Domain Fatigue Analysis
- Severity: Minor
- Description: The *TD OPTIONS keyword allows you to specify a scale factor to transform stresses from the Flexcom unit system to units consistent with the S-N curve data. However any specified scale factor is ignored by the program.
- Workaround in Flexcom 8.4.4: Generally no workaround is necessary, for a number of reasons. If you are using a Standard Unit System such as Metric (.keyxm input files) or Imperial (.keyxi input files), the program is fully aware of all the relevant units. Even if you are using User-Defined Units (.keyx input files), Flexcom is usually able to identify the relevant units based on the gravitational constant. In the unlikely event that a workaround is actually necessary, you can use the SCF input under the *FATIGUE DATA keyword to achieve the desired effect.

Issue 4: Time History of Water Surface Elevation

- Location: Time History of Water Surface Elevation
- Severity: Minor
- Description: There are two minor issues associated with this feature.
  - For input time histories with a non-zero mean, there can be a discrepancy between the user-defined input wave elevation time history, and the actual time history applied in the Flexcom simulation. This issue is further complicated by the use of ensembles (which greatly improves computational efficiency), as each ensemble can have a different mean value, and this can lead to step changes in water surfaces elevation at the intersection points between successive ensembles.
  - Although they are completely unrelated features, when you define wave loading via a time history, Flexcom automatically employs the Secant Stiffness model to any non-linear material elements in the model, even if the default/tangent approach is requested.
- Workaround in Flexcom 8.4.4: Suitable workarounds are proposed as follows.
  - Manually adjust the input water surface elevation in a spreadsheet before performing the Flexcom simulation. Separate adjustments will be required for each ensemble.
  - The issue is caused by an input variable which is inadvertently overwritten by another variable. While it may sound arbitrary, a simple workaround is to specify a single Reference Point Boundary Condition. If it is not possible to accommodate this additional boundary condition in the main model, you can add a separate/token element and constrain it appropriately.

    Contact software.support@woodgroup.com should you require any additional information regarding these workarounds.

Issue 5: Soil Modelling

- Location: Soil Modelling
- Severity: Minor
- Description: For added user convenience, Flexcom 8.4 introduced three built-in Soil Models which can generate P-y curves automatically. There are two minor issues associated with this feature.
  - The soft and stiff clay models provide an option to generate P-y curves for static or cyclical loading, with static loading assumed by default. If you opt for cyclic loading (LOADING=CYCLIC), your selection is actually ignored by the program, and P-y curves corresponding to static loading are generated regardless.
  - Although the models function correctly in a technical sense (with the exception noted above), there are a couple of glitches with the recognition of the new keywords in the user interface. Even if you correctly follow the guidelines provided with the *P-Y keyword, you may notice that some keyword entries are unrecognised by the user interface, and that some inputs

are not correctly transferred between the Keyword Editor and the Table Editor.

- Workaround in Flexcom 8.4.4: Suitable workarounds are proposed as follows.
  - Manually define the required P-y data (TYPE=USER) corresponding to cyclic loading, rather than using the automatically generated option (e.g. TYPE=SOFT CLAY, LOADING=CYCLIC).
  - Simply ignore any associated error messages in the user interface. Double check the program output file to ensure that the relevant inputs have been correctly interpreted by the finite element engine.

Issue 6: Aspect Ratio of Elements Created via Lines

- Location: Finite Element Mesh Generation
- Severity: Minor
- Description: The Lines modelling feature provides an automatic mesh creation facility which helps to expedite the model creation process. The meshing algorithm is designed to ensure that the ratio between the lengths of adjacent elements cannot exceed a certain value (e.g. 1.5 is the default ratio). During the meshing process, a line section with varying element lengths is sub-divided internally into three distinct zones (provided it is sufficiently long), namely a step-up section (element lengths gradually increase), a constant section (element lengths remain evenly sized), and a step-down section (element lengths gradually decrease). Where the line section is curved, meshing algorithm the constant zone actually contains one more element than intended. The net effect of this is that one element is slightly longer than intended and the adjacent/subsequent element is slightly shorter than intended, leading to a violation of the user-specified aspect ratio at that point in the model.
- Workaround in Flexcom 8.4.4: This is a minor issue and typically does not affect solution results in any meaningful way. However, should this pose a difficulty for any particular model, then it is possible to alter the specified aspect ratio or the suggested element lengths manually to compensate.

Issue 7: Application of Boundary Conditions on Non-Flat Rigid Seabed

- Location: Seabed Contact Modelling
- Severity: Minor
- Description: Rigid seabed contact is modelled in Flexcom by checking the positions of all nodes at every solution iteration. If a node is found to have penetrated the seabed, it is brought back to the mudline, and a boundary condition is applied in the direction normal to the surface to prevent further penetration. This operation is straight forward for flat seabed surfaces as the local seabed axis system coincides with the global axis system.

  This situation is a little more complex for sloping or arbitrary seabeds, where the solution axis system for any contact node is temporarily transformed to local, before responses are transferred back to the global system post-solution. A difficulty arises when a seabed node (e.g. the end of a pipeline) is assigned a constraint (i.e. via the *BOUNDARY keyword) in either DOF1 or DOF2, but not both. Flexcom cannot apply a constraint along an axis which does not coincide with the solution axis system (the seabed axis system in this case), so it simply constrains the node is both local DOFs in such circumstances. Two changes been made as part of this release in this area.
  - Where an additional constraint is applied, the user will be notified via an appropriate warning message for transparency.
  - Where the constrained node lies on a flat section of an arbitrary seabed, the additional constraint will not be applied, as it is not actually necessary. For example, this would allow the end of a line to slide horizontally along a flat section of seabed, subject to frictional constraints, if a global constraint was specified in DOF1.
- Workaround in Flexcom 8.4.4: The issue will rarely be encountered in reality, but where it does pose a problem, the obvious workaround is to use an elastic seabed model rather than a rigid one.

Issue 8: Exporting Plot Data to Excel

- Location: Plotting
- Severity: Minor
- Description: The Plotting facility provides useful options for exporting plot data to Microsoft Excel, in both XML and CSV formats. However, when a plot has associated units (e.g. force values in kN) which are different to the base units for that parameter (e.g. force values in N), the data is exported to Excel in base units but the column header suggests scaled/actual units.
- Workaround in Flexcom 8.4.4: Rather than exporting plots individually to Excel from the Plotting module, invoke the *OUTPUT_FILES keyword to ensure that all plot data is automatically exported to Excel during the creation of plots. There is no issue with units when data is exported in this manner.

Issue 9: Axial Stress Computations

- Location: Database Postprocessing, Summary Postprocessing
- Severity: Major
- Description: Physical element properties are defined in the *GEOMETRIC_SETS keyword, while effective structural properties for use in calculating stresses may be optionally defined in the *PROPERTIES keyword. Where values are specified via *PROPERTIES, they are intended to take precedence for the purposes of stress computations. However, axial stress computations use the default element properties, regardless of any values explicitly specified for the purposes of stress computations.
- Workaround in Flexcom 8.4.4: There is no workaround for this issue.

Issue 10: Large Number of Guide Surfaces

- Location: Guide Surface Contact Modelling
- Severity: Minor
- Description: If a model contains a large number of guide surfaces, specifically greater than the number of element sets in the model, then the analysis simulation can terminate prematurely without warning. The issue is caused by

incorrect dimensioning of an array variable in the source code.

- Workaround in Flexcom 8.4.4: Create additional element sets using *ELEMENT SETS such that the number of element sets exceeds the number of contact surfaces. This circumvents the problem, allowing the simulation to run successfully to completion.

Issue 11: Excel Variations

- Location: Spreadsheet Based Variations
- Severity: Minor
- Description: If you use the *EXCEL VARIATIONS keyword to instruct Flexcom to read input data from a spreadsheet, Flexcom could, in theory at least, pick up an invalid value (such as #N/A, #REF or #NAME) from the Excel data, and write out a meaningless Parameter to the generated keyword file. Flexcom 8.6.1 incorporates more rigorous checks to ensure that error messages from Excel are highlighted to the Flexcom user.
- Workaround in Flexcom 8.4.4: Manually rectify the problematic cells in Excel before generating spreadsheet-based variations in Flexcom. In rare circumstances, the error within Excel may not be obvious, and may represent a transient problem within Excel itself (e.g. macros not fully loaded). Should this happen, the keyword file generation process may be repeated in Flexcom if required to ensure correct keyword files are produced.

Issue 12: Nodal Acceleration Plots

- Location: Database Postprocessing
- Severity: Minor
- Description: If you request a time history plot of resultant nodal acceleration, via the *TIMETRACE keyword (DOF 8), the program will issue an unexpected error message and the requested plot will not be produced.
- Workaround in Flexcom 8.4.4: Request time history plots of nodal acceleration in the individual degrees of freedom (DOF 1, 2 & 3), and then manually compute the resultant value in a spreadsheet. Alternatively you could write some customised Database Access Routines.

Issue 13: Linearisation of Nonlinear Materials

- Location: *NONLINEAR MODEL
- Severity: Minor
- Description: The above keyword may be used to control the transition of material properties from an initial linear state to a fully nonlinear model between successive restart analyses, and the option is also an integral part of the Curvature Slippage feature. If a model contains beam elements only, the feature operates correctly as expected. However, if the model also contains other ancillary element types such as hinges, springs etc., material properties may not be assigned correctly to the relevant elements. For example, if an element set has associated nonlinear material properties, but you wish to temporarily model it as a linear material, any specified linearisation request may be ignored by the program.
- Workaround in Flexcom 8.4.4: It may be possible to use alternative modelling approaches for the special element types. For example, a hinge may be modelled using a beam element of low bending and torsional stiffness and high axial stiffness. A spring element may be simulated using a beam element with the relevant linear or non-linear axial stiffness, and low bending and torsional stiffness terms.

Issue 14: Hydrodynamic Coupling of Floating Bodies in the Frequency Domain

- Location: Frequency Domain Coupled Analysis, Hydrodynamic Coupling
- Severity: Major
- Description: It is not possible to model the effects of hydrodynamic coupling between adjacent floating bodies in a frequency domain analysis. If hydrodynamic coefficients are specified, the program issues an unexpected error message and terminates.
- Workaround in Flexcom 8.4.4: There is no workaround for this issue.

Issue 15: Continuation Restart Analyses

- Location: Restart Analysis
- Severity: Minor.
- Description: The vast majority of restart analyses involve the addition of new loads such as current or waves, or changes to boundary conditions, and in these situations you use the *New Loads or BCs* restart type under the *RESTART keyword. Occasionally, a restart is performed which is simply a continuation of the previous run. An example here might be where a regular wave analysis is found not to have reached steady state, and must be continued for further wave periods. In such circumstances, you can use either the *Continuation - New Database* option (where a new database is created) or the *Continuation – Append* option (where new data is appended to the existing data in a single database). The *Continuation – Append* option does not function correctly in Flexcom 8.4.4, and the software will issue an unexpected error message relating to 'incompatible databases'.
- Workaround in Flexcom 8.4.4: Use the *Continuation - New Database* restart option instead.

Issue 16: Combined Stress used in Fatigue Analysis

- Location: Time Domain Fatigue Analysis, Frequency Domain Fatigue Analysis
- Severity: Minor
- Description: When Flexcom is adding bending stress to axial stress to produce a combined stress at each fatigue hot spot, it uses an incorrect convention for the location of bending stress around the pipe circumference. Specifically,

  Flexcom measures the angle in 45° degree intervals in a clockwise manner from the local-y axis, when an anticlockwise convention should be used (for consistency with all other areas of the program). Predicted fatigue damage is still correct numerically, but the most critical location is presented as being 180° away from its actual location.

- Workaround in Flexcom 8.4.4: Generally no workaround is necessary, as the angular position is not typically of interest. The numerical value of the fatigue life at the most critical hotspot is correct, and this is the most crucial element.

Issue 17: Modal Analysis which includes Flex Joints

- Location: SHEAR7 Interface
- Severity: Major
- Description: When you perform a modal analysis of a structure which contains flex joints, the program can sometimes issue an unexpected error message and the requested SHEAR7 outputs will not be produced.
- Workaround in Flexcom 8.4.4: Although the issue happens rarely, there is no workaround when it does occur.