PERFORM-3D Version 5.0.0

Release Date: 2011-06-10

Contents

1. Installation Instructions for PERFORM-3D Version 5.0.0

The installation instructions are available in two places:

1. The "Installation Instructions" link on the PERFORM-3D CD browser.
2. In the file *PERFORM-3D_Install_Instructions.pdf* in the root folder of the CD.

The instructions include information about the software protection system used and installation options and instructions. You can choose between a Standalone or Network license.

For use of a Network license, see the *License Manager System Administrator's Guide* for more detailed information about the License Manager and the License Manager Administration program "WlmAdmin.exe".

This installation contains the Sentinel RMS License Manager 8.4.1. If you are currently running an earlier version of SentinelLM then it should be uninstalled prior to installing this version. ***IMPORTANT!*** All commuter licenses should be checked in before uninstalling the old version. If you are running a later version of Sentinel RMS License Manager then you do not need to install this one.

If you experience problems with the license please refer to the appropriate "*License Trouble Shooting Guide...*" located in the PERFORM-3D folder.

2. New Locations for Some Key Folders (IMPORTANT!)

If you are a new user you do not need to read this section.

PERFORM-3D makes use of a number of folders, for storing the executable program, the user manuals, ground acceleration records, and other things. For versions of PERFORM-3D before

Version 5.0.0, these folders are all located in the Microsoft Windows "Program Files" folder. Starting with Version 5.0.0, several of these folders have been relocated. The reason is that Microsoft has added security features to Windows that generally prevent the folders from being used in their old locations.

For an explanation of the new folder locations, see the following document in the Manuals folder:

Perform3D User Guide.pdf

3. Release Notes: Significant Changes from Version 4.0.4

Enhancements Implemented

- ***Incident 32611: 32 and 64 bit Versions.*** Version 5.0.0 has 32- and 64-bit versions of the analysis engine (the graphical interface is still 32-bit). When PERFORM-3D is installed on a 32-bit computer, it automatically uses the 32 bit engine, and on a 64 bit computer it uses the 64 bit engine.

  Test have shown that the 64-bit engine executes substantially faster than Version 4.0.4. The 32 bit engine may be significantly faster than Version 4.0.4.

  As with Version 4.0.4, the 32-bit engine has a limited capacity. If an analysis model is too large for the 32 bit engine, its size must be reduced. The 64 bit engine can analyze much larger models. However, users should recognize that a very large model can take a very long time to analyze. Many, if not most, analysis models are larger than they need to be (that is, they have more nodes and elements than are needed). For linear structural analysis, the computational penalty for using an unnecessarily large model may be acceptable. For inelastic analysis, however, the computational penalty can be very large.

  Users of earlier versions will see differences in the progress messages that are displayed during an analysis. Also, in Version 4.0.4, to stop an analysis it is necessary to position the cursor over the "Stop" button, but not press the button, and it may take some time for the analysis to stop. In Version 5.0.0 it is necessary to press the button, and the response will usually be much quicker.

- ***Incident 32612: Added Documentation, PERFORM-3DBinaryResultsFiles.*** For each analysis, PERFORM writes the analysis results in a number (often a large number) of short binary files. Many users access these binary files directly, to add post-processing operations that are not included in the PERFORM graphic interface.

  For earlier versions, the documentation of these files has been informal, and has not been included in the standard PERFORM documentation. For Version 5.0.0 these files are documented more formally, in the file PERFORMBinaryResultsFiles.pdf.

  It is necessary to write computer code to access the binary files. This is simple for most of the files, and almost any programming language can be used.

- ***Incident 32613: Large-capacity Eigenvalue Routine.*** In earlier versions, the eigenvalue routine could not accommodate structures with very large numbers of mass points. A different eigenvalue routine (the same as in SAP2000) is used in Version 5.0.0. This routine allows large numbers of mass points.

- ***Incident 31614: Stability Check.*** When an analysis model is set up for a complex structure, a common problem is that the model may be poorly conditioned numerically, or may be unstable. For an explanation of the causes of poor conditioning and instability, see the book Modeling for Structural Analysis: Behavior and Basics by Powell, Chapter 3, especially Section 3.5. This book is available from CSI.

  Poor conditioning affects the stiffness matrix. During equation solving, the stiffness matrix is "factorized", and the stiffness coefficients (especially the diagonal coefficients) get progressively smaller. In a well conditioned structure, a typical diagonal stiffness coefficient might change by 1 to 4 orders of magnitude. That is, a typical diagonal stiffness coefficient might get smaller by a factor of 10 to 10000. Numerically, this means that the difference between the logarithm (to the base 10) of the original stiffness coefficient and the coefficient after factorization is roughly 1 to 4 (see the book Static and Dynamic Analysis of Structures by Wilson, Appendix C, especially Section C.11).

  This can be termed the number of "lost" digits. If the stiffness matrix is well conditioned, the number of lost digits is roughly 1 to 4. Since the number of digits used in the numerical computations is about 15, this loss is not significant. However, if the stiffness matrix is poorly conditioned, the number of lost digits can be much larger. If this number exceeds about 12, there can be a severe loss of accuracy in the factorization. If a structure is unstable, one or more of the diagonal stiffness coefficients after factorization can, theoretically, become zero (for neutral stability) or negative (for instability). In this case the number of lost digits is theoretically infinite, and the factorization cannot be carried out. In practice, a stiffness coefficient will not be exactly zero, because of round-off in the calculation, but it can be very small. This is indicated when the number of lost digits exceeds about 14. In such cases, a factorized stiffness matrix may be obtained, but the result is meaningless.

  PERFORM Version 5.0.0 adds a feature to calculate the number of "lost" digits, show this number, and display a warning if the number is large. This is done in the stiffness matrix factorization phase for mode shape and period calculation. The number of lost digits is shown near the beginning of the M000.txt file. If this number is large, a warning message is displayed on the screen during the analysis, and is also printed in M000.txt.

  If PERFORM calculates a very small diagonal stiffness coefficient during factorization (i.e., if PERFORM detects instability) the execution does not stop. Instead, PERFORM replaces the coefficient with a value that corresponds to 12 "lost" digits. Usually (but not always) this will allow the stiffness matrix to be factorized. If the instability is of a global kind, the calculated periods of vibration will usually be very large. Sometimes they can be infinite (shown as "NaN" in M000.txt). If the instability is local, changing the diagonal stiffness coefficient may correct the instability, and the correct periods and mode shapes may be calculated. However, this may not always be the case.

In M000.txt, if the number of lost digits is large, PERFORM shows the DOF where the poor conditioning first appears. This may help to identify the cause of the poor conditioning or instability.

- ***Incident 31615: Printing of Mode Shapes, File M000.txt.*** When mode shapes and periods are calculated, the modal properties are printed in the text file M000.txt. In earlier versions, all mode shapes are printed in this file, and for a large structure the file can be very large. In Version 5.0.0, the number of mode shapes to be printed can be specified. Usually this number will be zero, so that detailed mode shapes are not printed, and the M000.txt file is quite short.

  For most structures there is no need to include the mode shapes in the M000.txt file. If the mode shapes are needed for post-processing calculations, it is easier to obtain them from the binary results files, rather than from M000.txt. For information on accessing the binary results files, see the documentation PERFORMBinaryResultsFiles.pdf.

- ***Incident 31616: Printing of Analysis Log, Files ECHO.txt and ECHOxxx.txt.*** The input data for a structure is printed in the ECHO.txt file, for each Analysis Series. In earlier versions, a "log" of each analysis in the series is also included in ECHO.txt, showing details of the analysis, the computation time for each analysis step, and an energy balance. For a large structure, this can make the ECHO.txt file very long. It can also be difficult to find the log for a particular analysis.

  In Version 5.0.0, the ECHO.txt file contains only the input data. The log for each analysis is in a separate file, ECHOxxx.txt, where "xxx" is the analysis number in the Analysis Series. For example, the first analysis is usually for gravity load, and the analysis log is the file ECHO001.txt.

Incidents Resolved

- ***Incident 31617: Adding 2-Node and 4-Node Deformation Gage Elements.*** An Incident was resolved in which it was not always possible in the user interface to add both 2-node gage elements (strain gages or beam type rotation gages) and 4-node gage elements (wall type rotation gages or shear strain gages) to the same model.

- ***Incident 31618: "Plot Loops" Option In The Component Properties Task.*** An incident was resolved where, in the Component Properties task, the hysteresis loops for rigid-plastic hinge components the loops may be plotted incorrectly. This affected only the Plot Loops option. The calculation of hysteretic behavior during analysis was not affected by this error.

- ***Incident 31619: Incorrect Units When Structure Section Results Are Saved To A Text File.*** An Incident was resolved in which the units shown for the time-history structure section forces saved to a text file could have been incorrect, even though the values themselves were correct. The units shown were those chosen for element results rather than structure-section results.

- ***Incident 31621: Possible Initialization Error.*** An Incident was resolved where a fatal error could occur when trying to open a model which had been first created during a PERFORM-3D session during which an existing model was previously open. This did not occur for models that were created after starting a new PERFORM-3D session without opening an existing model.

- ***Incident 31622: Possible Error When Importing Elements.*** An incident was resolved in which it was not possible to import elements into the first element group in the element group list. Instead an error message was generated. This problem did not occur if you imported elements into a group other than the first group.

- ***Incident 31623: Error When Using Only "Auto" Panel Zone Components.*** An incident was resolved in which an error message was generated when running the analysis for a model that had only "auto" panel zone components without any "regular" panel zone components present. This prevented the analysis from running.

- ***Incident 31624: Response Spectrum Analysis Results Are Not Saved Correctly.*** An incident was resolved in which the results for response spectrum analyses were not available after closing the program and then re-opening the model in a new session of Perform-3D. Response-spectrum results were available after running the analysis but before closing the program. No other types of analysis results were affected.

- ***Incident 31625: PMM Hinge Component with Both Tri-linear Behavior and Energy Degradation.*** An incident was resolved in which the analysis would sometimes terminate with a message saying that convergence could not be obtained in the case of a structure having column elements with PMM hinge components when one or more of those components had both (1) a trilinear F-D relationship and (2) energy degradation. The message would refer to the state calculation for a CH3A or SH3A component.

- ***Incident 31626: Self Weight Loads Incorrect for Infill Panel Elements.*** An incident was resolved in which the self-weight loads for the Infill Panel elements were not correct. One quarter of the element weight should have been applied at each of the four element nodes. However, the load was only being applied at the element "I" node, and this load was one quarter of the expected element weight.

- ***Incident 31627: Incorrect Colors for BRB Elements.*** An incident was resolved in which the reported D/C ratios in compression for BRB elements could be incorrect in color-coded Deflected Shape plots and in the text files that are written from the Deflected Shapes task. This problem could allow the compression D/C ratio for a BRB element in a deflected shape plot to decrease, whereas it should increase monotonically. The details are as follows: At each step in the analysis, for each BRB element, PERFORM calculates deformation D/C ratios for (a) tension deformation and (b) compression deformation. If the tension D/C ratio in the current step is larger than the previous maximum value, the maximum value is updated. This was done correctly, and tension D/C ratios never decreased. If the compression D/C ratio in the current step is larger than the previous maximum value, the maximum value should similarly be updated. However, the current compression D/C ratio was incorrectly compared with the previous tension maximum, not the compression maximum. If the previous tension maximum

was smaller than the previous compression maximum, the compression D/C ratio can decrease. The error affected only color-coded deflected shape plots. The largest D/C ratio reached for any element was correct (i.e., the "highest" color that any element reaches was correct). However, there could have been a change to a "lower" color, in which case that color was not correct. The effect was minor if the maximum usage ratio in tension was close to that in compression. It will be usual to consider usage ratios at the end of an earthquake. In the Deflected Shape task, it was possible for the D/C ratio for an element at the end of an earthquake to be smaller than the correct maximum. For static push-over analysis, usage ratios typically increase monotonically, and the error should have had no effect.

- ***Incident 31628: Can Not Specify Strength Loss for "Auto" Panel Zone Components.*** An incident was resolved in which strength loss could be specified for "Auto" Panel Zone components. An error message was generated and the specification was rejected.

- ***Incident 31629: Incorrect Time-History Results for Frame elements.*** An incident was resolved for the Time History task in which the signs of the bending moments about Axis 2 were incorrect (positive values are shown with a negative sign, and vice versa) for results at the element ends of frame elements.