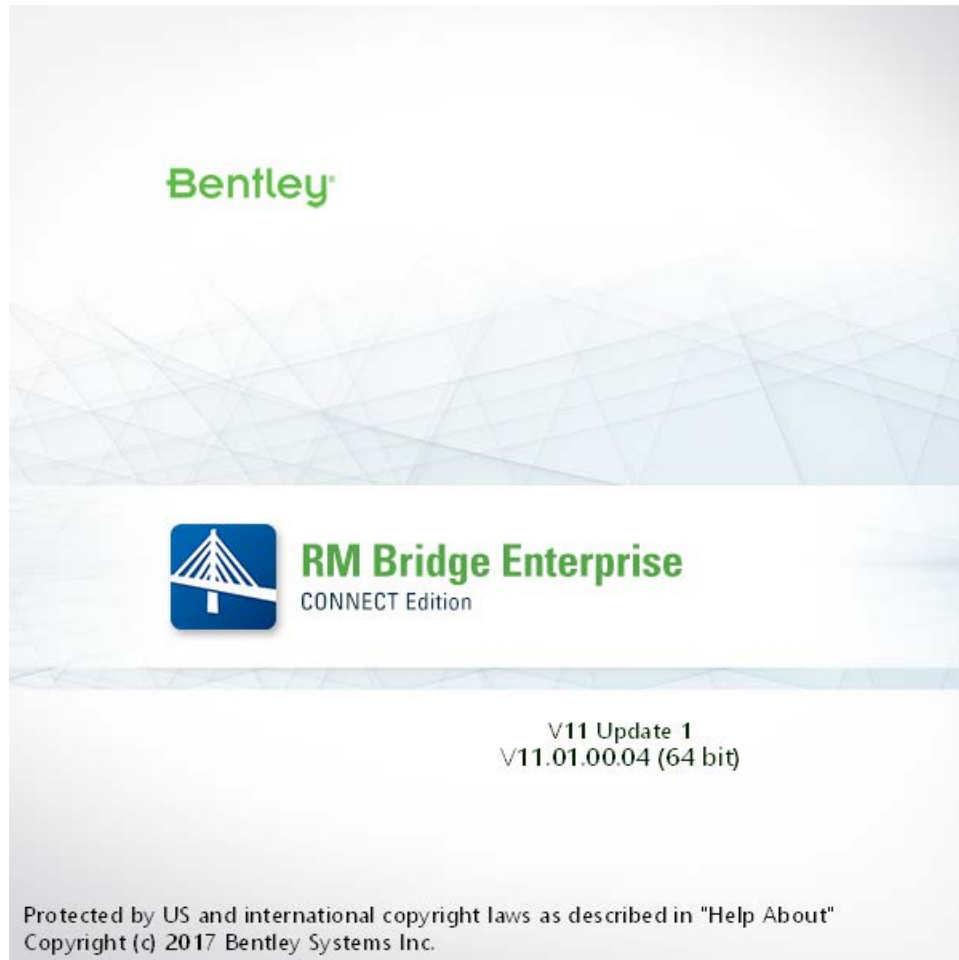




RM Bridge CONNECT Edition V11 Update 1 Release Notes



Enhancements in RM Bridge CONNECT Edition V11 Update 1 (V11.01.00):

RM Bridge CONNECT Edition V11 Update 1 contains the following enhancements and error corrections:

1. RM 64-bit
2. Parallel Sparse Solver
3. FEM: Enhancements for Shell Elements
 - Envelope-FE Results for Shell Elements
 - Export Load Case-FE and Envelope-FE Result to Excel

- FE Result integration for Composite Sections
- Quantity Report for FEM
- FE Cross-section Modeling Improvements
- 4. Enhancements in GUI including:
 - Assign load in main window graphically,
 - New substructure wizard in Modeler,
 - Add graphical support in Analyzer,
 - Copy/paste node coordinate data from RM to excel and vice-versa,
 - Automatic node and element numbering for creating new models in Modeler,
 - Copy/paste variable Table data from excel to RM and vice-versa,
 - Direct access to RM-Set (RHIST) in Tint action (time history analysis),
 - Results View,
 - Display element, nodes, and analysis time count
- 5. Enhanced TCL Editor: Error/Warning list generation, and WinMerge/Diff Control
- 6. Other Enhancements
 - New live load macro for South Africa standard,
 - User Damping and mode selection for dynamic wind analysis,
 - Excel file output for influence lines and List Sup files,
 - TCL command extensions to access eigen value analysis results,
 - Stress leading superposition results output is now added to list sup,
 - Trapezoidal load applied from begin to end of element series
- 7. RM View: Quantity report and Automatic import of generated plot files in RM Bridge
- 8. Error Corrections

Note: RM Bridge View automatically will be installed with RM Bridge, RM Bridge Advanced, and RM Bridge Enterprise on your machine. Prior to installing the latest version, please uninstall any pre-existing RM Bridge View.

If you get an installation error message 1603 or 2738, please follow these steps:

1. Click on the Start menu, choose Run, type cmd and click OK
2. For 32-bit OS
 - To unregister the VBScript engine, run this command: reg delete "HKCU\SOFTWARE\Classes\CLSID\{B54F3741-5B07-11CF-A4B0-00AA004A55E8}" /f
 - To unregister the JScript engine, run this command: reg delete "HKCU\SOFTWARE\Classes\CLSID\{F414C260-6AC0-11CF-B6D1-00AA00BBBB58}" /f
3. 64-bit OS
 - To unregister the VBScript engine, run this command: reg delete "HKCU\SOFTWARE\Classes\Wow6432Node\CLSID\{B54F3741-5B07-11CF-A4B0-00AA004A55E8}" /f
 - To unregister the JScript engine, run this command: reg delete "HKCU\SOFTWARE\Classes\Wow6432Node\CLSID\{F414C260-6AC0-11CF-B6D1-00AA00BBBB58}" /f
4. Retry installation

1. RM 64-bit

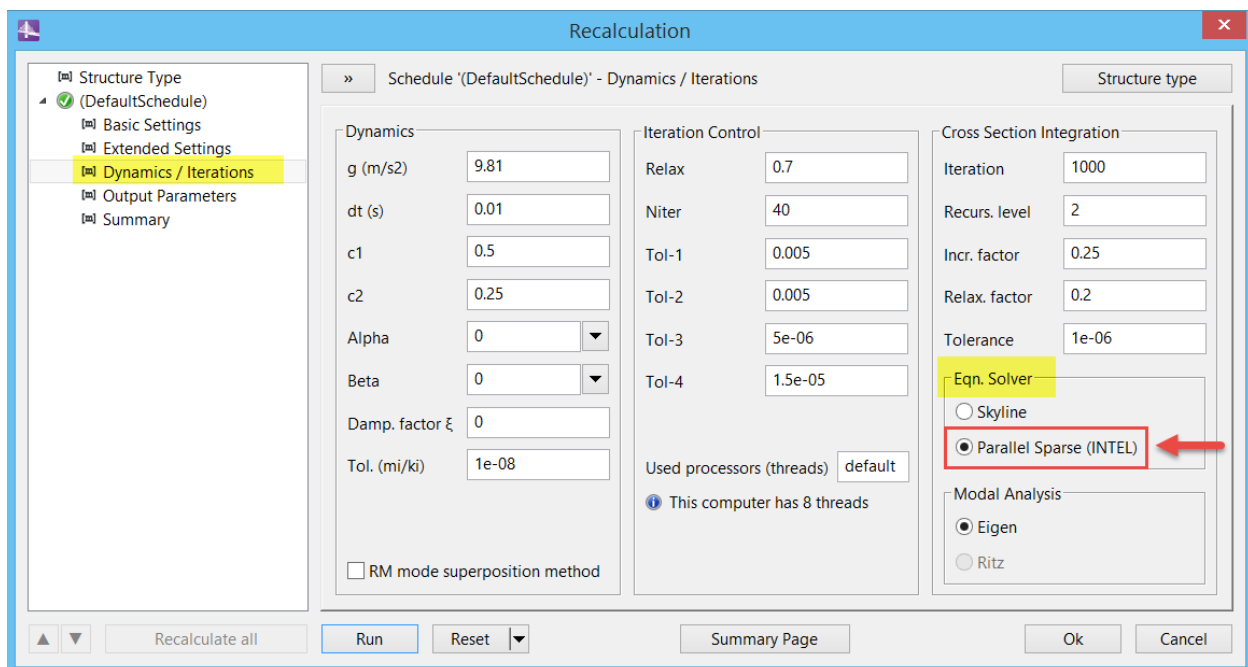
A new 64-bit version of the software is now available. This version allows for the execution of very large models and significant increase in the speed of analysis as well.

Note: The difference in machine precision between 32-bit and 64-bit may result in numerical difference in results generated by the two versions of the program. It should not however result in significant difference in resulting design from the two analyses.

2. Parallel Sparse Solver

The new solver which improve the calculation speed for dynamic analysis has been added. User can choose between the old skyline solver or the new intel parallel solver for faster calculations on repetitive analysis projects like time history, live load analysis, AddCon, etc.

This new solver can be found under Dynamic/Iterations under Eqn. Solver as it shown below:



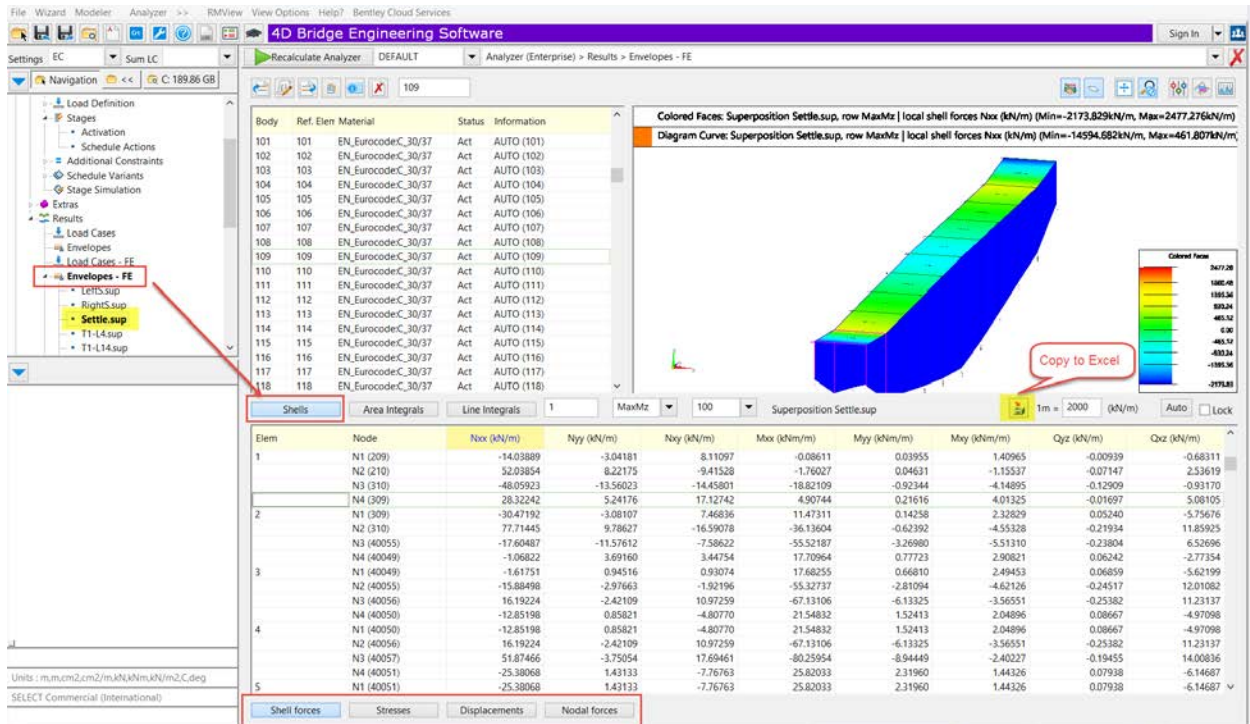
A new tcl command for parallel solver is also introduced. This new tcl command is listed below:

SOESOL SKYLINE/INTEL

3. FEM: Enhancements for Shell Elements

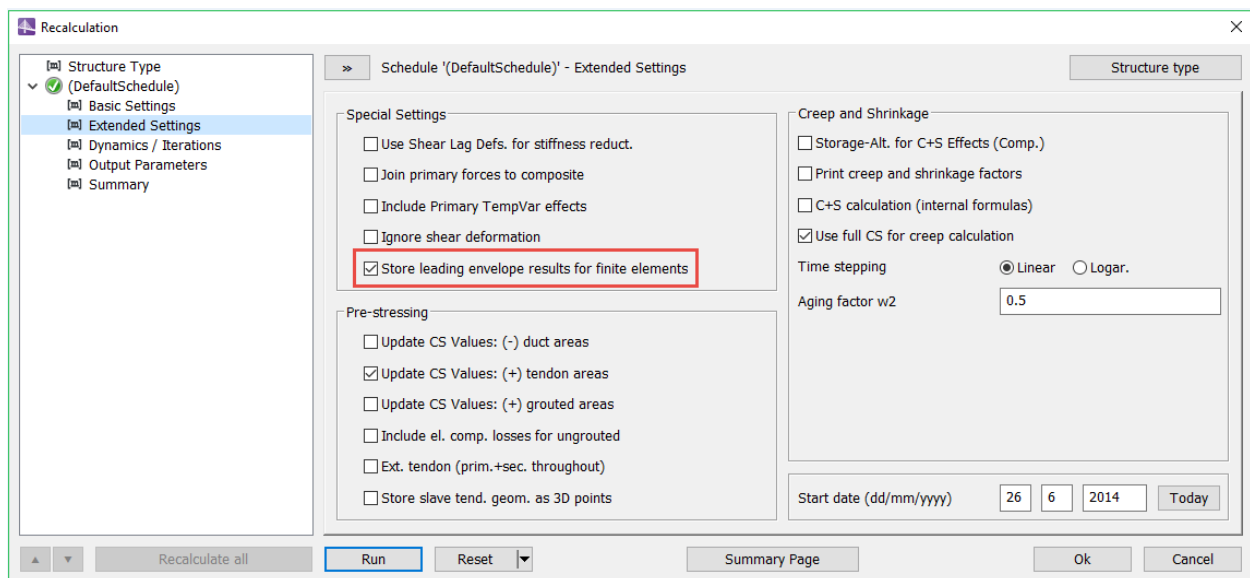
3.1 Envelope-FE Results for Shell Elements:

An option to report the Shell forces, Stresses, Displacements, and Nodal forces for Shells under the Envelopes-FE is now available. The Shell results for Envelope-FE can be viewed as shown below:



Note: The results presented for Envelopes-FE are available only for load cases superposition. To access these results in the superposition file, user needs to enable “Store leading envelopes results for finite elements” in the Recalculation pad in Analyzer under Extended Setting as it shown below:

Envelope-FE results for shell elements will be disabled with blank output if user don’t select this option.



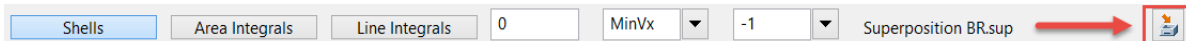
Note: In the current release, FEM envelope results for Shell elements in the Result pad are only displayed for area integral 100 (first defined area integral).

3.2 Export Load Case-FE and Envelope-FE Result to Excel:

A new button for exporting load case-FE and envelope-FE is now available.




Copy to Clipboard: Select all data by using Ctrl+A or Hold Shift and click on desired rows to select different data. Click on “Copy to clipboard” icon or use Ctrl+C to copy the selected data then use Ctrl+V to paste into an excel file.



3.3 FE Result integration for Composite Sections:

The composite section integral results for finite element models can be obtained by specifying the join option in the load case result. This requires a beam like composite definition to be made for the finite element cross-sections as well. In cases, where composite element is activated the integral results are shown as normal results like it is presented for beams.

3.4 Quantity Report for FEM:

Quantity report is now available for FE models. User needs to calculate the model then click on the Quantity Icon  in the main window.

3.5 FE Cross-section Modeling Improvements:

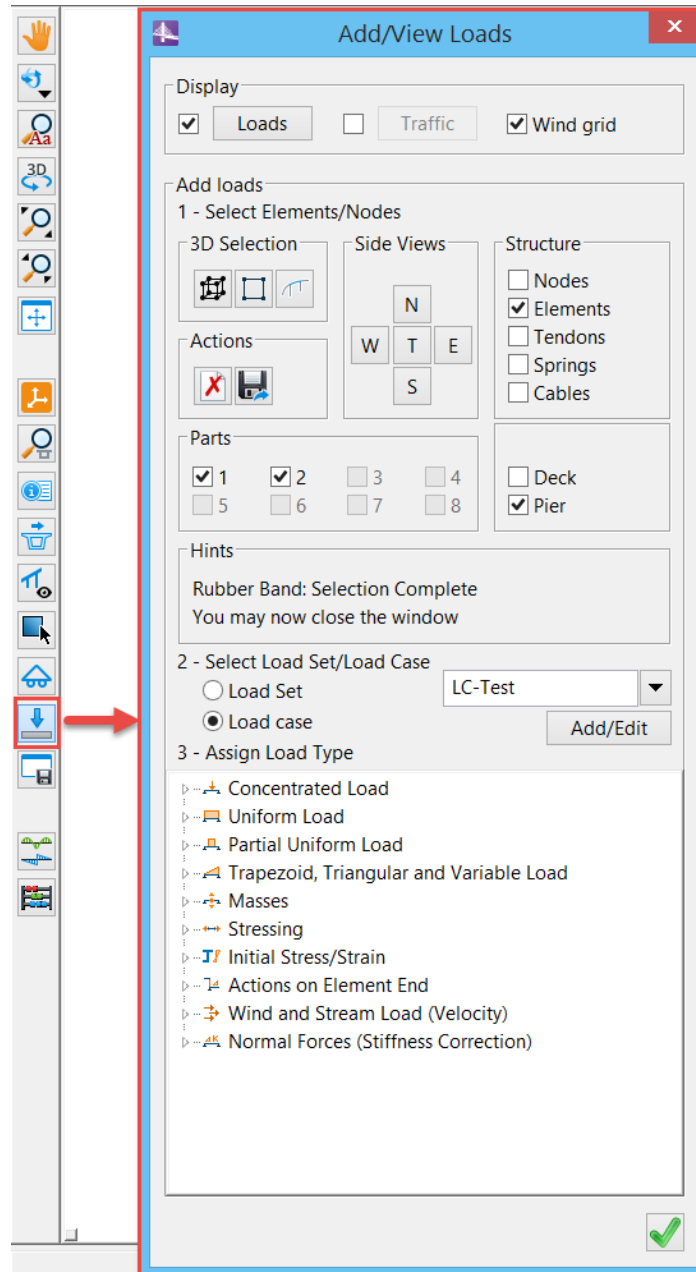
- User no more needs to redefine the part number while generating the shell elements. The shell element automatically determines the part it belongs to from the cross-section finite element it is laid upon.
- The part assigned to shell elements always lie on the nearest intersection of the construction lines.
- A warning is added if a finite element is laid over two different cross-section finite elements i.e. different part numbers.
- A correction is made to adjust the thickness of the shell element according to the cross-section element it is laid upon.
- A new hint message screen is added on the top right-hand side of the cross-section window for better viewing of the hints.
- Creation of area integral is now done in one less step.

4. Enhancements in GUI

Several GUI enhancements have been implemented to improve the user experience. They include:

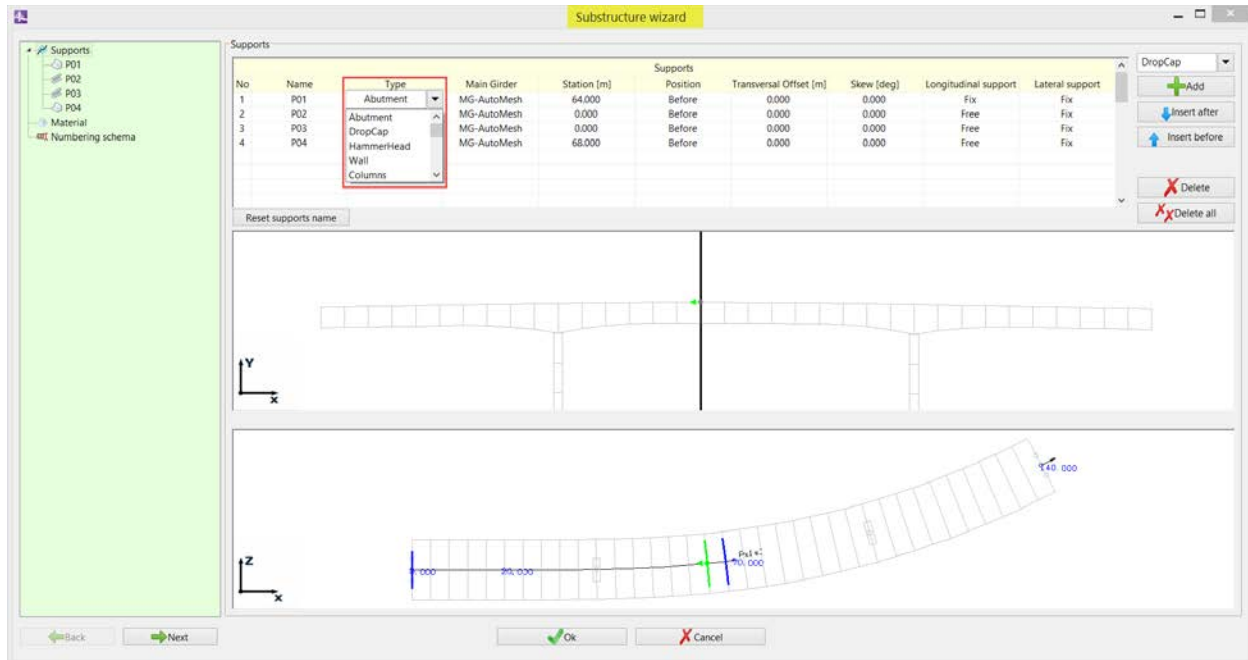
4.1 Assign Loads in The Main Window Graphically:

View load icon in main window has been extended to Add/View Loads which allow user to create new load sets or load cases using 3D-graphical selection in the main window. User can also view traffic loads and wind grid.



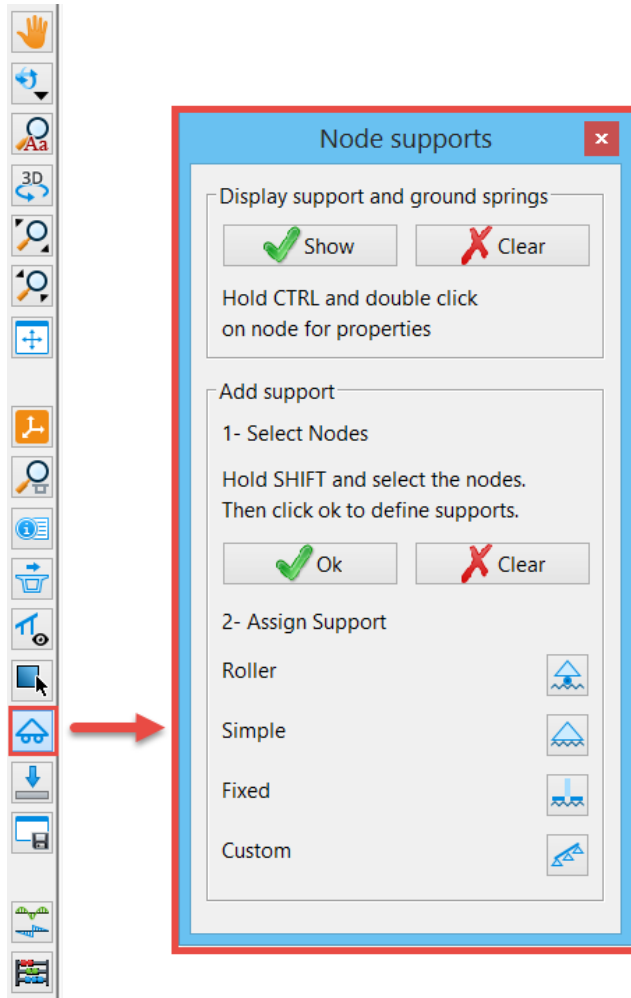
4.2 New Substructure Wizard in Modeler:

A new option for creating substructures in Modeler have been added. This new option allows to define multiple supports at once. The new feature is available by right click on “Piers” tree item inside Modeler and click on “Wizard Substructure”. There are different substructures which can be define including: Abutment, Drop Cap Pier, Hammer Head, Wall, and Columns.





4.3 Add Graphical Support in Analyzer:

A new option for creating node supports graphically in Analyzer have been added. This new option allows to define multiple supports at once. Existing node supports and any ground spring connection can also be viewed.



4.4 Copy/paste Node Coordinate Data from RM to Excel and Vise-versa:

A new option for copy/paste node coordinates from/to Excel has been included. Data can be copied to excel (clipboard) using  and can be pasted from excel (clipboard) using .

| Node | X | Y | Z | Status | Support | Wind Grid | Diagram Outp | Node | X | Y | Z | Status | Support | Wind Grid | Diagram Outp |
|------|--------|-------|-------|--------|---------|-----------|--------------|------|---------|---------|--------|--------|---------|-----------|--------------|
| 101 | 0.000 | 0.000 | 0.000 | Act | No | | | 1301 | 99.296 | -24.559 | 8.036 | Act | No | | |
| 102 | 4.000 | 0.067 | 0.000 | Act | No | | | 1302 | 99.296 | -19.559 | 8.036 | Act | No | | |
| 103 | 8.000 | 0.133 | 0.000 | Act | No | | | 1303 | 99.296 | -14.559 | 8.036 | Act | No | | |
| 104 | 12.000 | 0.200 | 0.000 | Act | No | | | 1304 | 99.296 | -9.559 | 8.036 | Act | No | | |
| 105 | 16.000 | 0.267 | 0.000 | Act | No | | | 1400 | 136.453 | -3.792 | 22.662 | Act | No | | |
| 106 | 20.000 | 0.333 | 0.000 | Act | No | | | | | | | | | | |
| 107 | 24.000 | 0.400 | 0.001 | Act | No | | | | | | | | | | |
| 108 | 28.000 | 0.467 | 0.005 | Act | No | | | | | | | | | | |
| 109 | 32.000 | 0.533 | 0.025 | Act | No | | | | | | | | | | |
| 110 | 36.000 | 0.599 | 0.068 | Act | No | | | | | | | | | | |
| 111 | 39.999 | 0.641 | 0.133 | Act | No | | | | | | | | | | |
| 112 | 43.998 | 0.684 | 0.239 | Act | No | | | | | | | | | | |
| 113 | 47.996 | 0.719 | 0.366 | Act | No | | | | | | | | | | |
| 114 | 51.992 | 0.745 | 0.546 | Act | No | | | | | | | | | | |
| 115 | 55.985 | 0.764 | 0.777 | Act | No | | | | | | | | | | |
| 116 | 59.974 | 0.775 | 1.066 | Act | No | | | | | | | | | | |

4.5 Automatic Numbering for Creating New Models:

The screenshot shows the 'Numbering schema' dialog box with the following data:

| | Start | Step | Offset | Next Start |
|------------|-------|------|--------|------------|
| Element | 101 | 1 | 100 | 401 |
| Node Start | 101 | 1 | 100 | 201 |
| Node End | 102 | 1 | 100 | 202 |
| Spring | 5101 | 1 | 10 | 5101 |

| | Start | Step | Offset | Next Start |
|------------|-------|------|--------|------------|
| Element | 10101 | 1 | 100 | 10101 |
| Node Start | 10101 | 1 | 100 | 10101 |
| Node End | 10102 | 1 | 100 | 10102 |
| Spring | 15101 | 1 | 10 | 15101 |

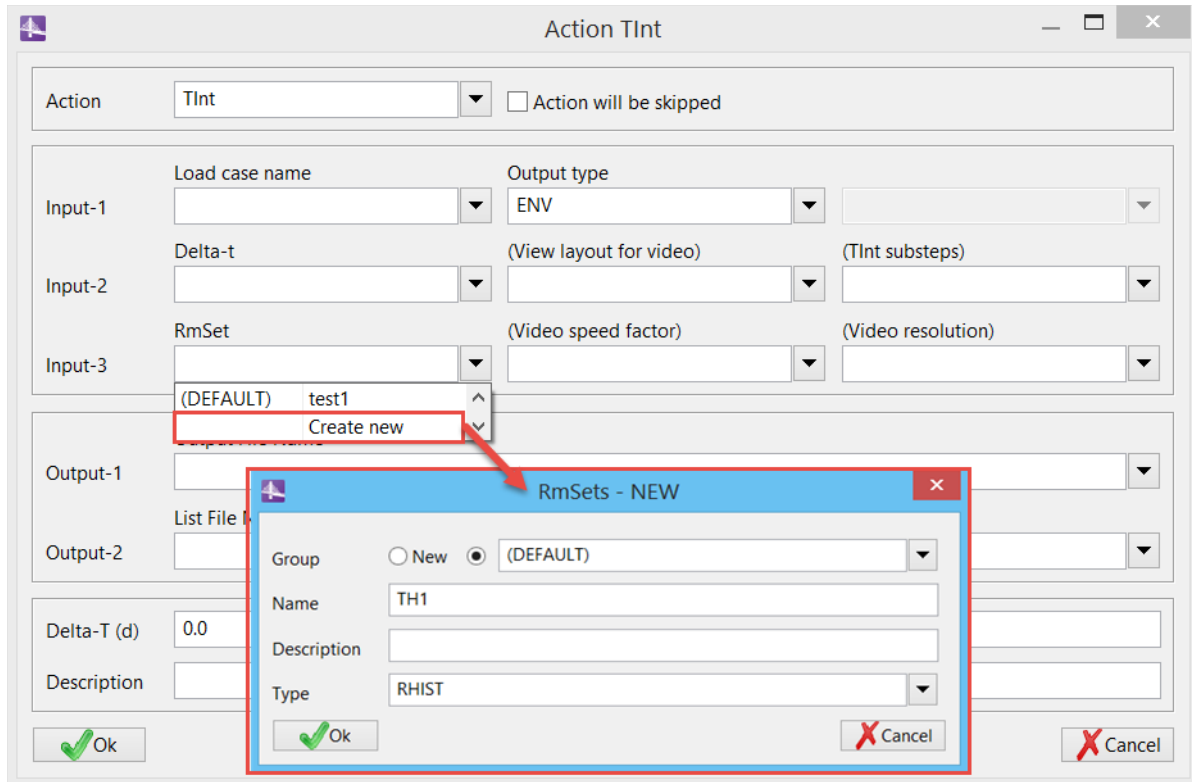
| | Start | Step | Offset | Next Start |
|------------|-------|------|--------|------------|
| Element | 20101 | 1 | 100 | 20101 |
| Node Start | 20101 | 1 | 100 | 20101 |
| Node End | 20102 | 1 | 100 | 20102 |
| Spring | 25101 | 1 | 10 | 25101 |

| | Start | Step | Offset | Next Start |
|------------|-------|------|--------|------------|
| Element | 30101 | 1 | 100 | 30101 |
| Node Start | 30101 | 1 | 100 | 30101 |
| Node End | 30102 | 1 | 100 | 30102 |
| Spring | 35101 | 1 | 10 | 35101 |

New buttons for copy/paste data in variable "Table" from/to Excel is now available. Data can



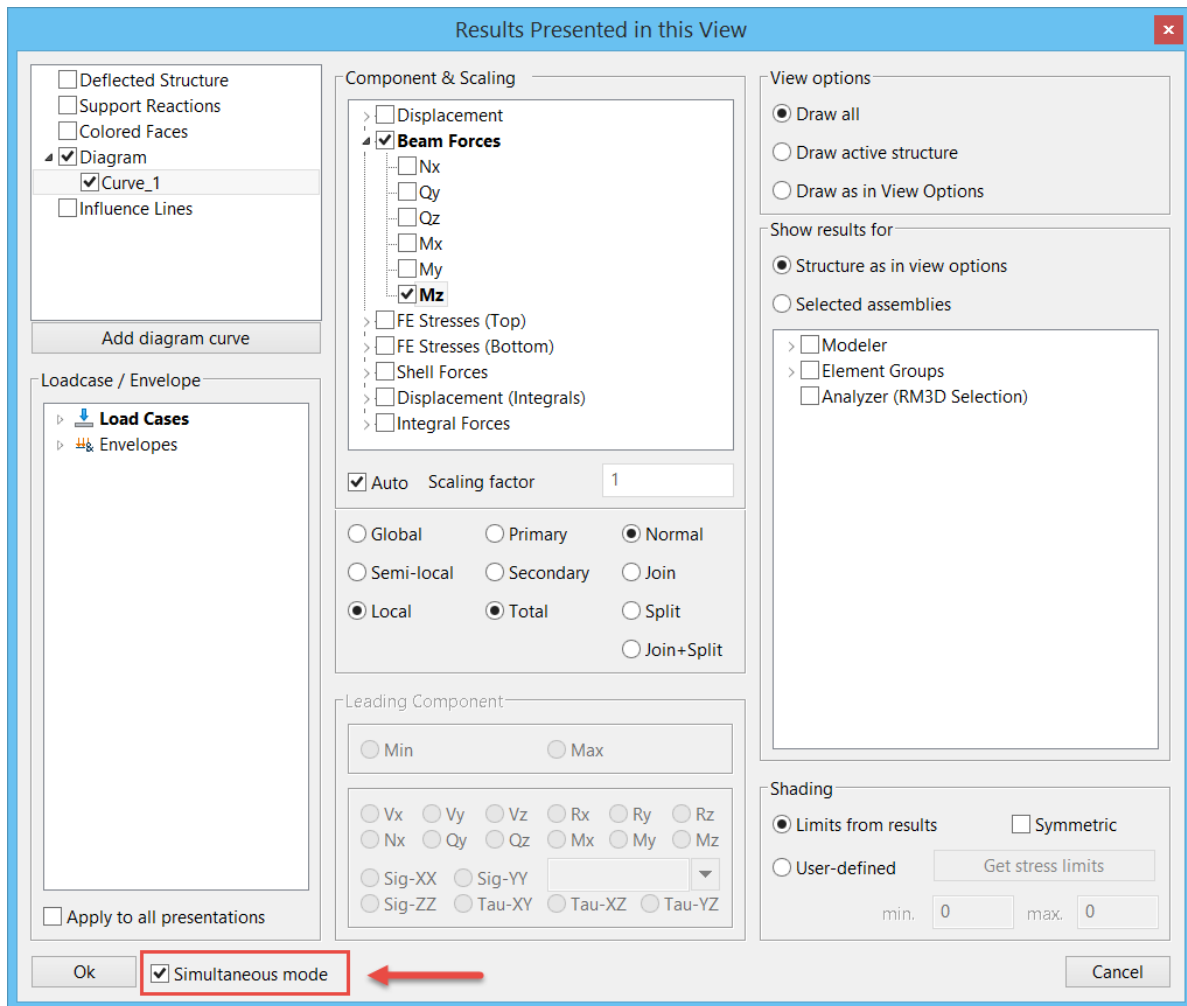
A new option for creating a new RM-Set (RHIST) directly from Action Tint for time history analysis is now available.



4.8 Results View

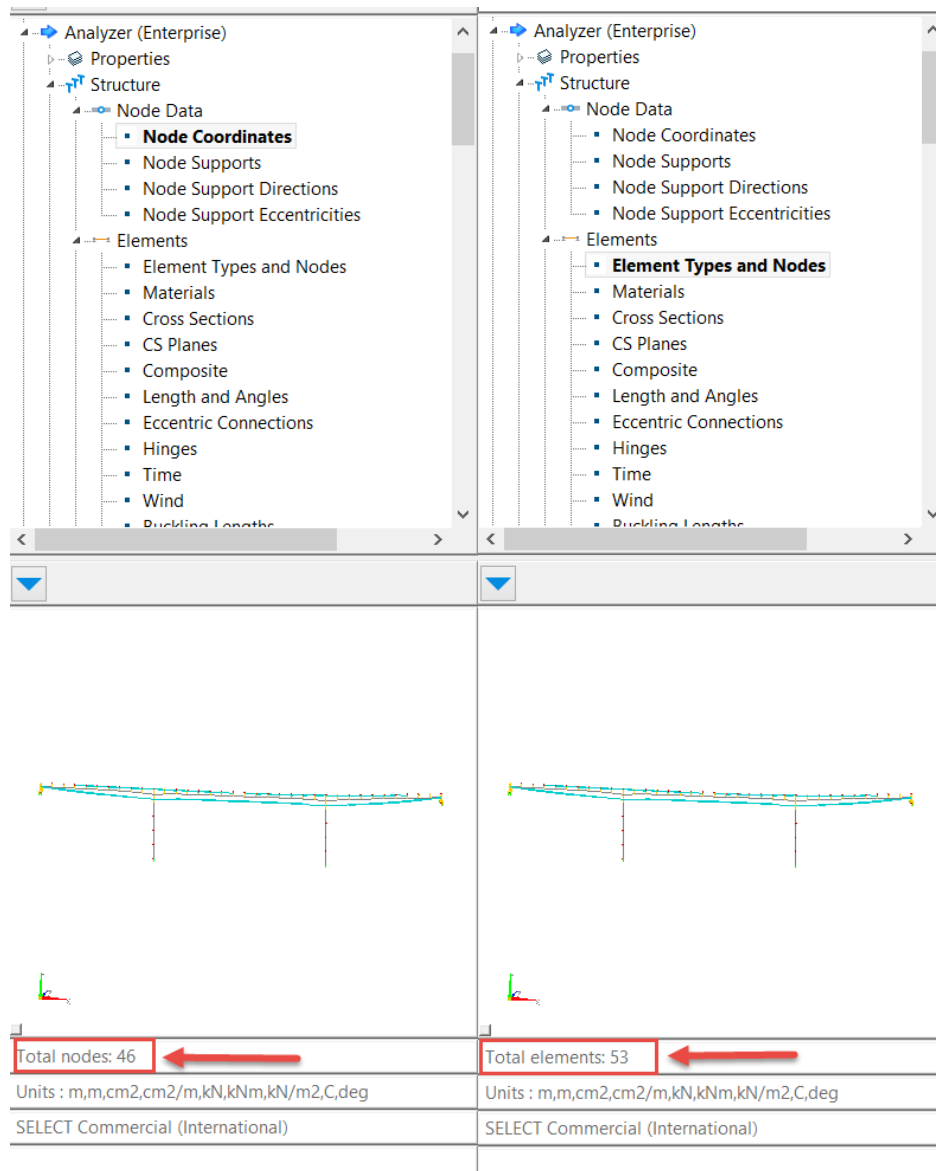
Simultaneous results view in the Results pad is now available. Results display in main window get updated automatically by clicking on any result options.

Also, results dialog box in the tree has been updated. For more information click on F1-Help.

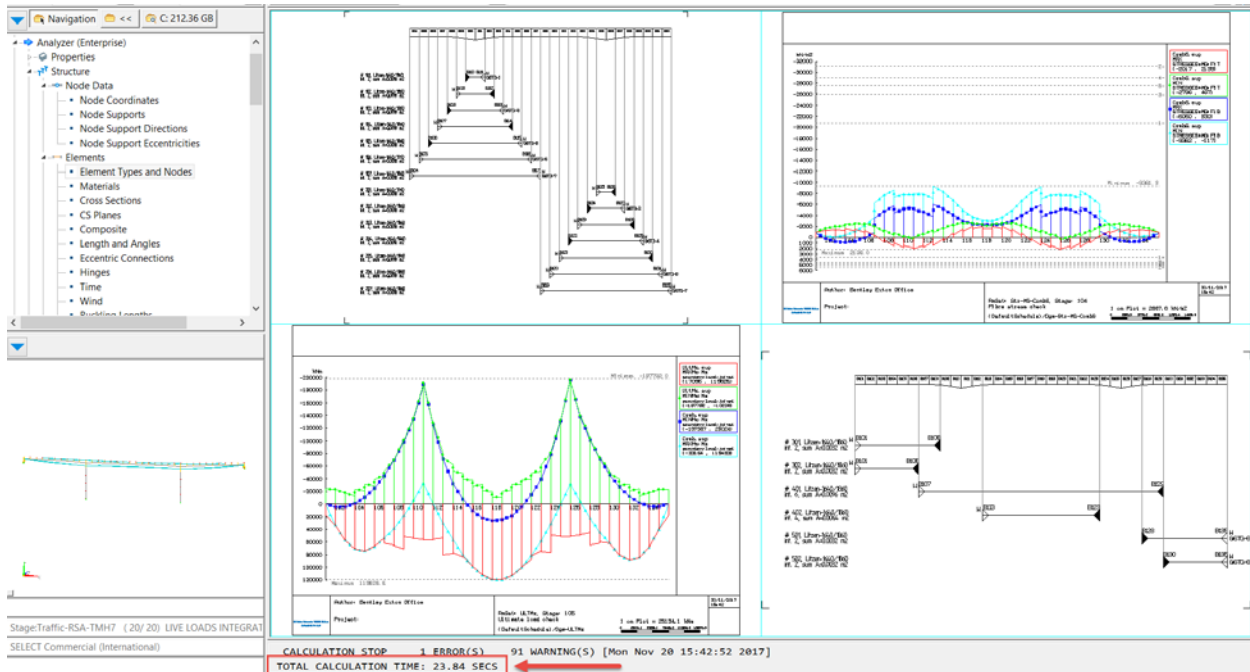


4.9 Display Number of Element, Nodes, and Analysis Time Count

The information about total number of elements and nodes is now display in the left bottom panel.

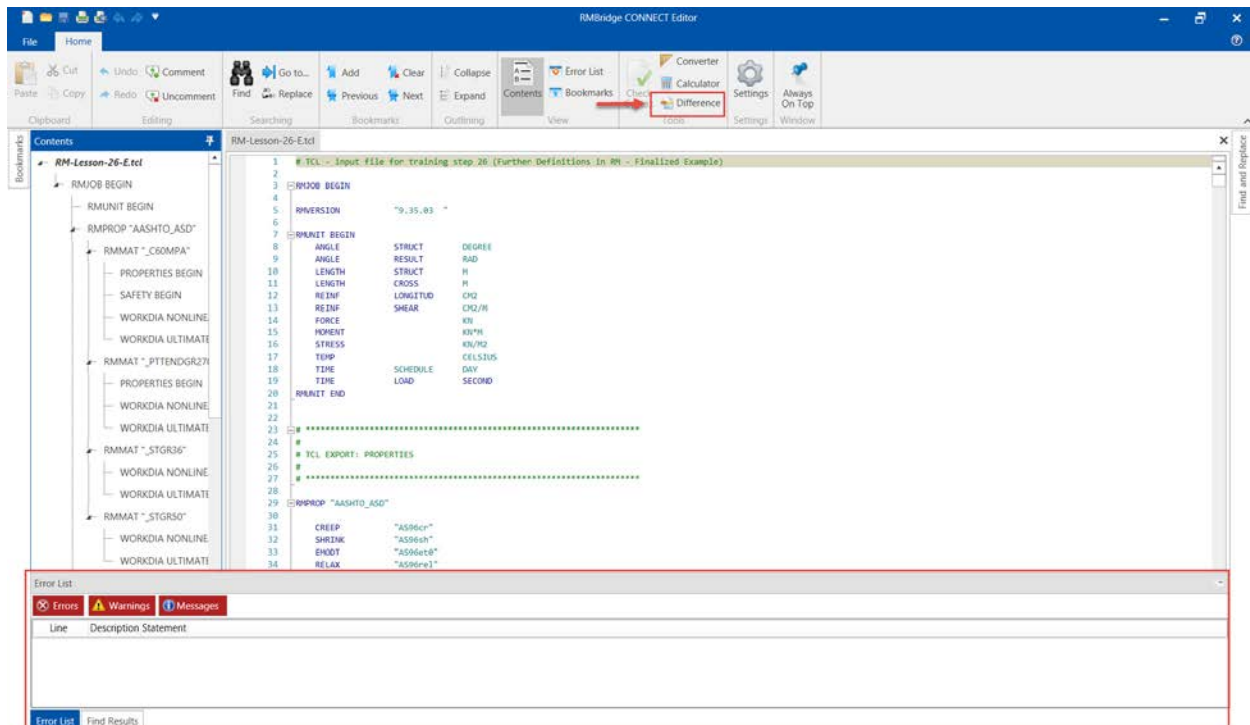


Total analysis time count is now display in the bottom of main window for user info.



5. Enhanced TCL Editor

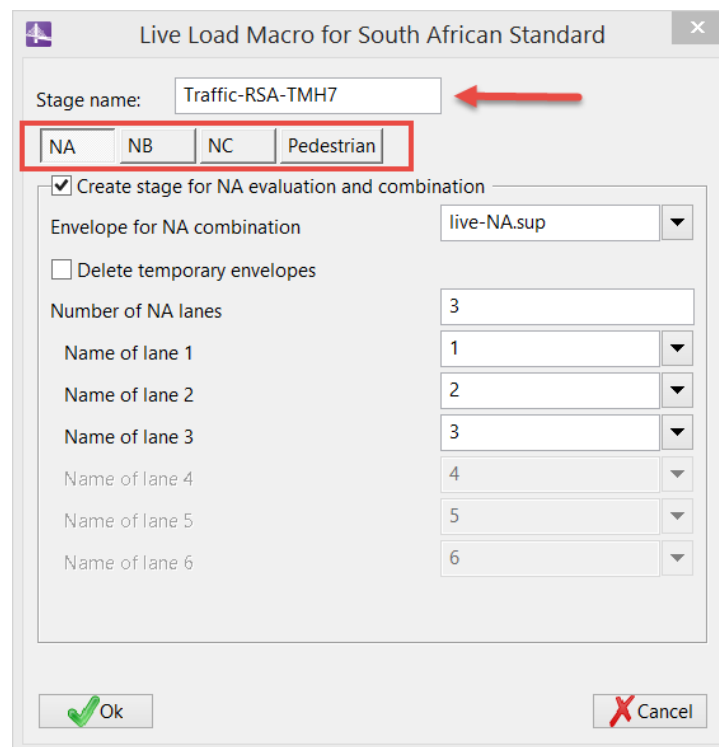
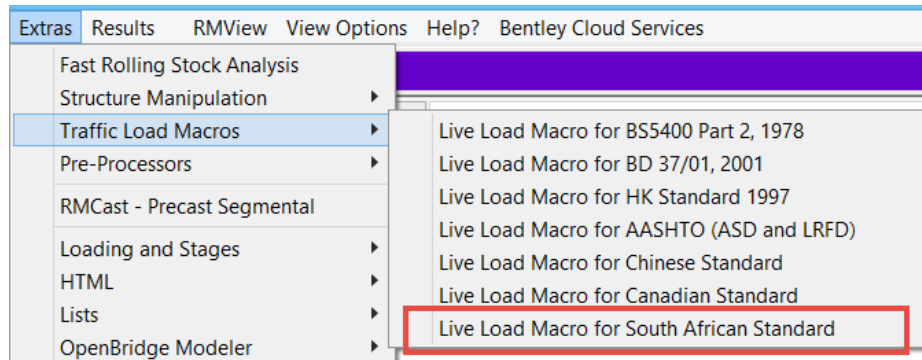
Enhanced TCL Editor is set as default editor. “Error list” for viewing errors/warnings in tcl scripts and “Differences” for comparing tcl scripts are now available.



6. Other Enhancements

6.1 New Live Load Macro Per South African Standard (TMH7):

A new live load macro per South African Standard (TMH7) is now available. It includes combinations for NA, NB, NC, and pedestrian loading. It can be found in Traffic Load Macros under Extras.



Note: The load trains for South Africa can be imported to the project from Extras → Loading and Stages → Load Train Definition → South African Standard

Load Train - South African Standard, TMH7 ...

The template will be imported into the current project.

☒ NA
 ☒ NB
 ☒ NC
 ☒ Pedestrian

Width of pedestrian loading (m)

☒ Append load trains after existing definitions

Options

☒ Read to RM Database
☐ Save to TCL file

6.2 User Damping and Mode Selection for Dynamic Wind Analysis

New inputs for user defined damping and mode selection have been added for Wind Analysis (Action Wind). This new input is optional and when provided is considered in the analysis as well. Click on F1-Help for additional information about these new inputs.

Action Wind

Action ☐ Action will be skipped

Input-1 Wind number (Type of damp. dependency) (Damping table name)

Input-2 Damping constant Dx:Dy:Dz

Input-3 Modal file name ALL or ACTIVE (RM-Set)

Output-1 Superposition file name

Output-2 List file

Delta-T (Day)

Description

6.3 Excel file output for influence lines (ListInf) and List Sup (ListSup)

Excel file output for actions ListInf and ListSup are now available.

The screenshot shows the 'Action ListInf' dialog box. The 'Action' dropdown is set to 'ListInf'. There is an unchecked checkbox labeled 'Action will be skipped'. The 'Influence file name' section has three empty input fields. Below this are 'Input-1', 'Input-2', and 'Input-3', each with an empty input field and a dropdown arrow. The 'Output-1' section is labeled '(Excel file)' and has an empty input field with a dropdown arrow. The 'Output-2' section is labeled 'List file' and has an input field containing an asterisk (*). The 'Delta-T (Day)' field is set to '0.0'. The 'Description' field is empty. At the bottom, there are 'Ok' and 'Cancel' buttons.

Action ListSup ☐ Action will be skipped

Superposition file name

Input-1

(Split || Join)

Input-2

(EXP)

Input-3

Output-1 (Excel file)

Output-2 List file

Delta-T (Day)

Description

6.4 TCL Command Extensions to Access Eigen Value Analysis Results

New TCL commands to access Eigenvalues for elements and nodes by mode shapes are now available. Please see section 2.8.38 in the TCL script guide for more information.

6.5 Stress Leading Superposition Results Output

Stress leading superposition results output is now added to list sup in the Result tree and RM-Sets.

6.6 Trapezoidal Load Applied from Begin to End of Element Series

A new option which allows to apply the trapezoidal load from begin to end of element series is now available.

Trapezoidal Element Load - TG

Elements

☒ Element series

From: 101

To: 101

Step: 1

☐ Assembly

☐ Distribute load from beg -> end elem node

Element Begin

Qxb (kN/m): 0

Qyb (kN/m): 0

Qzb (kN/m): 0

Element End

Qxe (kN/m): 0

Qye (kN/m): 0

Qze (kN/m): 0

☒ Real length

☐ Projection

☐ Nodal load

☒ Load per length unit

☐ Load mult. by CS width

☐ Load mult. by CS depth

☐ User area (m-m)

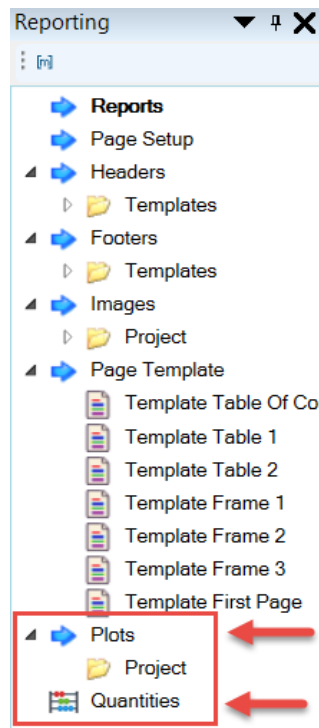
Dy: 0.0000 Dz: 0.0000

☒ Global ☐ Local

Ok Ok+New Explanation Cancel

7. RM View Enhancements

Quantity report including cost estimate and plots generated in RM are now available in RM View. For more information click on F1-Help in RM View Report.



8. Error Corrections

The following errors have been corrected:

- The program used incorrect grouted area for pretensioning tendons when user selected “Update CS values: (+) grouted area” in the extended setting under recalculation pad. This may have resulted in incorrect forces on structural members. This has been fixed by changing the grouted area for pretension tendons equal to zero.
- FE Computation at Gauss Points: The plate element formulation was reporting stresses which were directly calculated at the node points. In this version, an improvement is made to calculate stresses at gauss points and then they are extrapolated to node points. This approach produces more accurate stresses.
- The issue of wrong Reactions presentation in the Main View have been fixed.
- The program accidentally replaced wrong numbers when using two noses in ILM. Also, the tendon elements generated were not well assignment. These issues have been corrected.
- Sometimes, two-part numbers at the same position would be generated for finite elements laying over the same parts. This was not expected behavior and is now corrected to generate the same part number in such situations.
- The program didn’t show the assembly selection in the new load case when user creates this new load case using "copy the current line". This issue has been corrected.
- The program was not considered the haunch in the formulation for Hollow box bridge wizard for AASHTO code. This issue has been corrected.
- The program used incorrect value for tensile strength of prestressing steel in calculation of relaxation per AASHTO ASD method. This has been corrected by assigning correct values in the material property.
- PushoverD analysis: The program used incorrect value for the ground acceleration equal to 9.81 m/s/s for both US Customary and SI units. This has been corrected for converting spectral diagram from SI units to US units to find the performance point in Pushover analysis.
- The issue with using assemblies in ILM has been fixed.

Product Interoperability

- OpenBridge Modeler
- MicroStation, GEOPAK, InRoads, MXROAD

System Requirements

Supported Operating Systems:

- Windows 7, 8, 10 (32 and 64 bit)

Prerequisites:

- Windows Installer Version 3.1v2
- Microsoft .NET Framework 3.5 SP1
- Microsoft Access Runtime 2007

Processor:

- Intel or AMD processor 2.0 GHz or greater

Memory:

- 8 GB minimum, 32GB or higher recommended

Hard Disk:

- 700 MB free disk space (includes 300 MB install footprint for a complete installation)

Video:

- 250 MB of video RAM or higher recommended

Quick Install Guide

To install, run the installation .exe file and follow the prompts. All prerequisites will be installed as required. A 64-bit RM Bridge will install under “Program Files\Bentley\RM Bridge” by default.