Latest October 2023 – Updates/Upgrades to Vespa/Vespa2 MSE Design Software. Item 1. Design Methodology

Vespa now includes AASHTO 2010 LRFD and AASHTO 2015 LRFD Design Methodologies. (Jan 2016)

1.1 AASHTO 2010 LRFD - Update/Upgrades:

1.1.1 Static Analysis:

1.1.1.1 Live/Dead Loads

The original version of AASHTO 2010 LRFD included with Vespa treated Live Loads as an "equivalent height of soil" in accordance with 11.10.10.2 (i.e. 600mm of soil depth).

In an effort to model loads more explicitly, this method was replaced with the ability to set the actual surcharge load (kN/sq.m or lbf/sq.ft), load type (Live/Dead), and offset from the face of the Wall.

When loading older Vespa files, the equivalent Live Load will be automatically calculated based on the previous input equivalent soil height. The other result of this change is the load factor applied to the Live Load. Modelled as a soil surcharge (previous version), a Load Factor of 1.5 (ES)was applied. When modelled as an actual Live Load Surcharge (current version), a Load Factor 1.75 is applied.

1.1.1.2 Block/Grid Connection Factors

Previously, Reduction Factors for Creep (RFcr) and Durability (RFd) at the Block/Grid Connection were defaulted to be those provided by the Geogrid manufacturer for Creep and Durability of the Geogrid Reinforcement. These were editable, however, if they were changed, they would alter the Geogrid Reinforcement properties as well as the Connection Properties. New fields have been provided to separate these Reduction Factors for a) Reinforcement and b) Connection.

AASHTO

Tult	RFcr	RFd	LTDS
52.55 kN/m	1.55	1.10	26.80 kN/m
	RFid	Cds	Ci
20mm- gravels or aggrega ⌄	1.15	0.90	0.90
			Alpha Correction
			0.80

Connection Properties

Min. Connection Capacity
7.00 kN/m

Normal Load (IP-1)	Normal Load (IP-2)
38.00 kN/m	53.00 kN/m

Conn. Capacity (IP-1)	Max. Conn Capacity (IP-2)
27.00 kN/m	29.60 kN/m

RFcn-Creep	RFcn-Durability	Tlot Reduction
1.00	1.00	1.00

Shear with Reinforcement

Initial Shear	Shear Angle	Maximum Shear
13.70 kN/m	57.00 °	89.00 kN/m

Generation Increment
0.30 m

Reset to Defaults

1.1.1.3 Seismic

Previous AASHTO 2010 did not include Live Load in the Seismic analysis. It is now included with the 0.5 combined Load Factor.

1.2 AASHTO 2015-LRFD

1.2.1 Static Analysis

1.2.1.1 Tmax Calculation

Per Figure 11.10.6.2.1.2, for calculating the Vertical Pressure on reinforcements in sloping situations, the extent of the triangular weight above the reinforced zone is limited to 0.7H as shown. This was previously (2010) modeled as the full depth of reinforcement (L). The 0.7H limitation is more conservative but better reflects the actual vertical stress applied to the reinforcement.

1.2.1.2 Pullout Resistance

2015 provides a more detailed explanation of the calculation of Vertical Confining Pressure and Zp depth in the anchorage zone beneath sloping backfill. Refer to Figure 11.10.6.3.2-1.
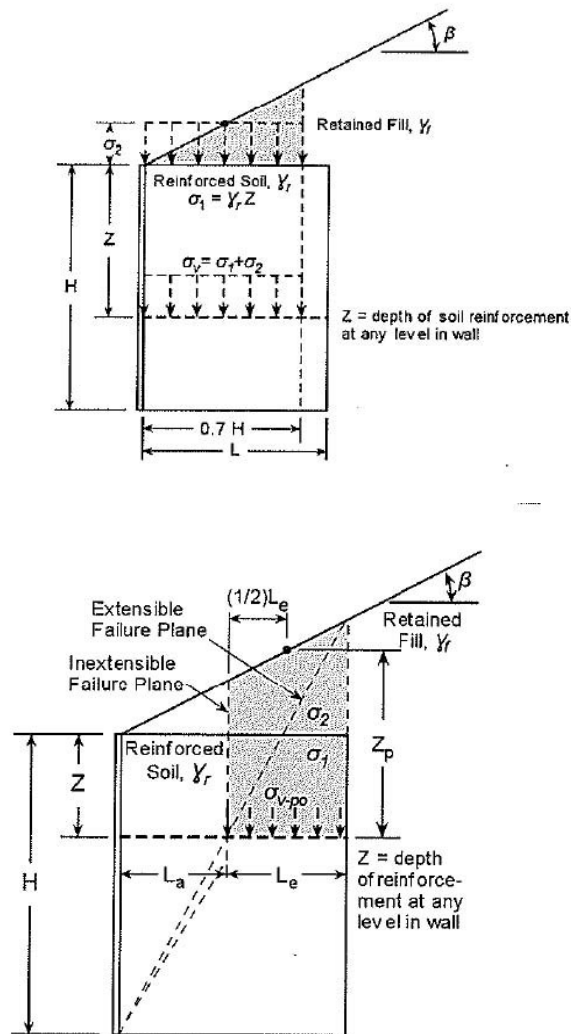
Nominal Vertical Confining Pressure:

$$\sigma_1 = \gamma_r Z$$

$$\sigma_2 = \gamma_f (Z_p - Z) \text{ for sloping backfill}$$

$$\sigma_{v-po} = \gamma_r Z + \gamma_f (Z_p - Z)$$

$$Z_p = Z + (L_a + (1/2)L_e) \tan \beta \text{ for sloping backfill}$$

1.2.1.32 Crest Toppling

Previously (2010), as the option was not provided to offset Live Loads from the back of the Wall, the Crest Toppling Calculation always included the Live Load surcharge.

With the provision for Live Load and Dead Load offsets, the Crest Toppling calculation now checks if the Load is outside of the zone of influence, assumed to be 1H:1V from the back of the Coping Unit.

1.2.2 Seismic Analysis

1.2.2.1 Calculation of Seismic Horizontal Acceleration Coefficient

Previously, the Seismic Horizontal Acceleration Coefficient, kh, was calculated as a function of the PGA and a specified allowable displacement. In 2015, the displacement criteria has been simplified to either allow it, or not, rather than specifying a specific amount of displacement. If displacement is allowed, the Seismic Horizontal Acceleration Coefficient, kh, is reducedto half of As. Refer to 11.10.7 for full Seismic Calculation.

1.2.2.2 Total Factored Horizontal Thrust (C 11.6.5.1)

As an additional check for the Total Horizontal Thrust, two potential scenarios are compared. Check 1 is 100% of Dynamic Horizontal Thrust due to Earth Pressure/Live/Dead Loads with 50% of the Inertial Force of the Reinforced Mass. Check 2 is 50% of the Pae/Paell/Paedl with 100% of the Inertial Force.

1.2.2.3 –Factored Increment Dynamic Inertia Force at Each Layer of Geogrid.

For 2015, Internal Stability, the total Inertial Force of the active wedge is evenly distributed to all reinforcements by simply dividing the Inertial Force, Pi, by the total number of Layers of Reinforcement. This is a change from 2010, where the distribution of the load to the Reinforcements was based on the Reinforcements relative contribution to total Anchorage Length.

1.2.2.4 – Pullout Calculations (Jan 2016 -end)

Previously, the Tmax was recalculated for Pullout in order to remove the Live Loads per 11.10.6.3.2-1 as the Live Load was not considered in the Pullout Resistance. In discussion with Designers and Vespa users, the concern was highlighted that for high walls, the case may occur when a live load (traffic lane) could exist in the "active" part of the wall, without necessarily have the same load above the anchorage zone. For conservatism, we have there used the same Tmax for Tensile Overstress and Pullout calculations.

1.2.2.5 – Dead Load Default Factors (Jan 2017)

For Dead Loads, Vespa always used the Default Load Factor value of 1.25 based on Table 3.4.1-2, where DC is described as Component and Attachments. The assumption here was that Dead Loads would typically be a footing for a structure nearby, or even an abutment footing. It has been suggested by user that the default for Dead Loads should be the more conservative ES: Earth Surcharge of 1.5, and let people reduce as they see fit, rather than the other way around. To be conservative, we now have the Dead Load default as 1.5.

Item 2. Project Tab

2.1 Creation of Project Folder Path

Previously, when the user entered the Client Name and Project Name, Vespa automatically created a File Folder Path using these names in the directory. Vespa users indicated that in most cases, they had already created their own Project Folder with related projects documents (CAD files, PDFs, emails, etc), so this automated process was not efficient. This process has been changed as follows:

•   Prior to "Creating a Wall" the user must select "Set/Change" in the Folder Path to identify the Project Folder Path. Using this method, an existing Project Directory can be selected and set to the

directory that Vespa uses to send files. If your Project files are usually found in the same place, you can set your starting pointing "Settings / Options / Project File Location". By setting your Project File location here (Settings), when you are in the Project Tab and select "Set/Change", it will automatically start you in the main directory.

- Once the Project Folder Path has been selected, you can now "Create Wall" as before.

Item 3 – Design Criteria

3.1 – Load and Resistance Factors (LRFD Methodology).

The Load and Resistance factors are now being formatted in a table showing the Min. and Max. "default" values, as well as the Min. and Max. "Used" values. Previously, for each load or resistance factor, a separate variable was designated for each Min.and Max value. We now have a single variable (i.e.LFES – Earth Surcharge Load Factor), with a min. and max. value depending on where itis being used. This is more in line with how itis presented in AASHTO(applies to 2010 and 2015, Static and Seismic.) For AASHTO 2010 and 2015 greater flexibility has been added to the Seismic Resistance Factors, allowing individual variables for each of the Connection, Pullout, and Tensile (Combined Static/Seismic) as well Frictional Resistance Factors for Connection and Pullout (RFconn, RFpull). Note that for RFconn, the default value is 1.0, however AASHTO recommends for "frictional" connections this be reduced to 0.8.

Design Notes | Empirical Checks | Load & Resistance Factors | Design Inputs

○ Static   ● Seismic

Term	Name	Min Def.	Max Def.	Min Used	Max Used
LFDC	Loac - Dead Load (Structure)	0.90	1.25	0.90	1.25
LFEQDL	Loac - Dead Load Seismic Factor	0.00	1.00	0.00	1.00
LFES	Loac - Earth Surcharge Load	0.75	1.50	0.75	1.50
LFEQ	Loac - Earthquake Load	0.00	1.00	0.00	1.00
LFEH	Loac - Horz. Pressure of earth fill	0.90	1.50	0.90	1.50
LFEQLL	Loac - Live Load Seismic Factor	0.00	0.50	0.00	0.50
LFEV	Loac - Vert. Pressure of earth fill	1.00	1.35	1.00	1.35
BEARING	Resistance - Bearing	1.00	n/a	1.00	n/a
COMBIMPCO	Resistance - Combined Static/EQ Connection	1.20	n/a	1.20	n/a
COMBIMPPO	Resistance - Combined Static/EQ Pullout	1.20	n/a	1.20	n/a
COMBIMPTO	Resistance - Combined Static/EQ Tensile	1.20	n/a	1.20	n/a
RFconn	Resistance - Seismic Frictional Connection Factor	0.80	n/a	0.80	n/a
RFpull	Resistance - Seismic Frictional Pullout Factor	0.80	n/a	0.80	n/a
SLIDING	Resistance - Sliding	1.00	n/a	1.00	n/a

Item 4 – Wall System

4.1 – Additional Wall System Characteristics

Additional fields have been incorporated to allow greater control over quantity calculation, panelization of units, etc.

4.1.1 Pins/Unit – For systems that require pins or connectors, this input will calculate the total quantity required for the project. If the manufacturer has not yet updated this field, it will be editable and the user can input a value as required.

4.1.2 Facia Batter – As a number of different systems have 2 or 3 possible facia batters for the same system, we have incorporated the ability to select just a single block, then select the facia batter from a drop down menu (if multiple batters exist). This is a change from the previous method where there existeda separate block for each wall batter(i.e. Block A ,8 degree, Block A,Vertical, etc).

4.1.3 Base Course – Some systems include a special base block, or base course, which is used exclusively on the first course of the Wall. This unit can now be defined and used in the panelization, quantity estimate, and analysis.

Item 5 – Reinforcement Tab

5.1 – RFcn-Creep and RFcn-Durablity

As discussed above, for AASHTO Design Methodologies, 2 additional inputs have been included to separate values for RFcn-Creep and Durablity. These reduction values relate to the Creep and Durability reduction at the Block/Grid connection.

5.2 –Generation Increment

The generation increment is a new feature that allows the Designer to set the Geogrid Length increment at which the "Generate"function (Design Tab) steps through a Trial Design. For example, if the Generation Increment was set to 1.0ft, the Generate function would step through possible Designs starting at the min. base to height ratio (i.e. 6.0ft on a 10.ft high Wall for NCMA), then keep stepping up by 1.0ft (7.0ft, 8.0ft, etc), until the Design met all required F.S. or CDR values.

 Item 6 – Stations Tab

6.1 Paste Function. As some users create the Wall Stations in Excel or some other spreadsheet, we now have the ability to paste cells from a spreadsheet directly into the Stations Table.
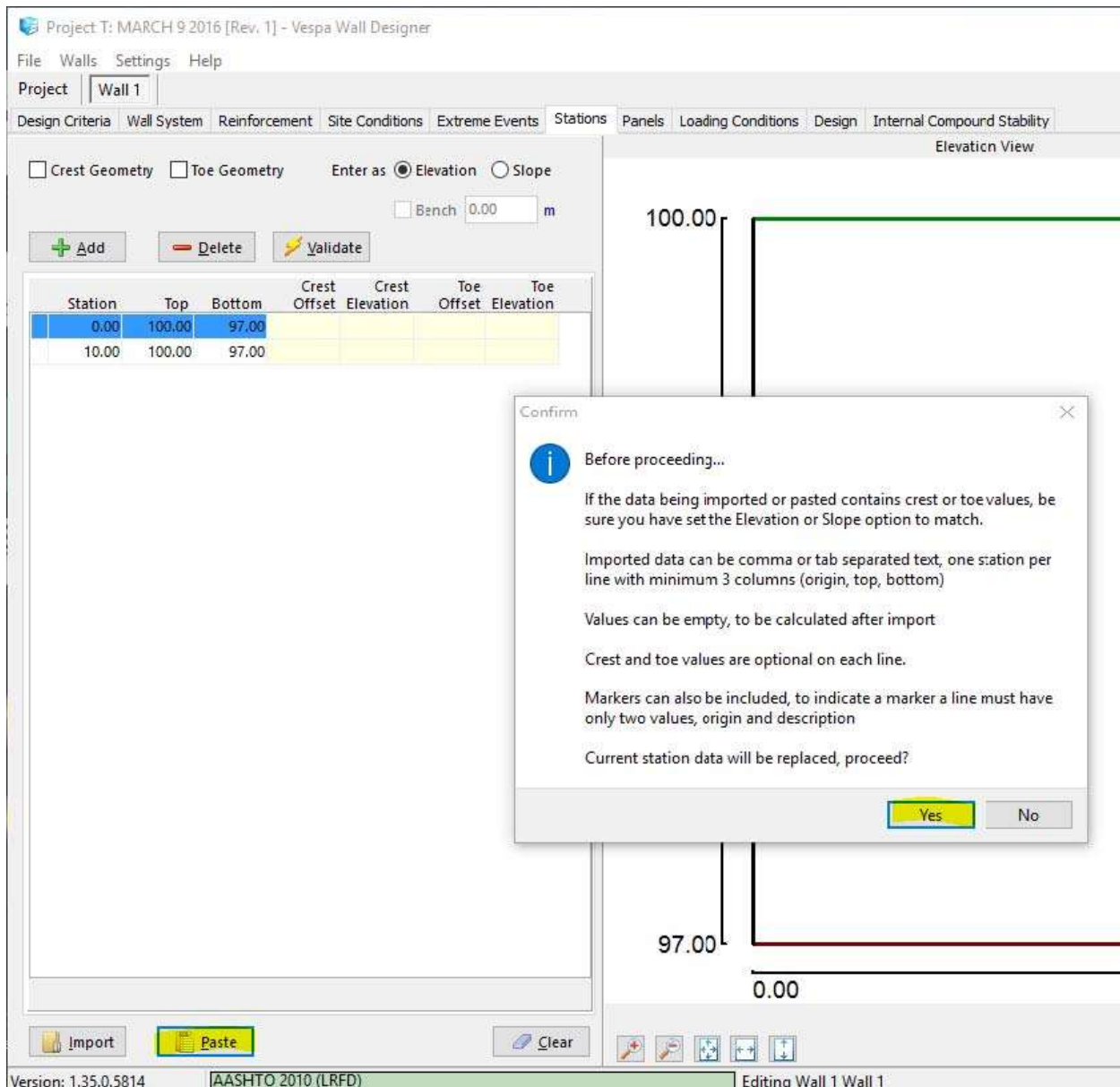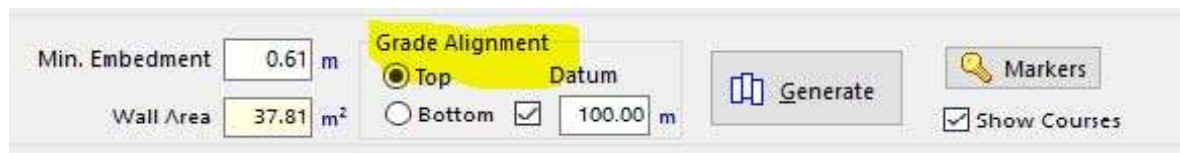
Item 7 – Panels Tab

A new feature has been added to allow the user to align either the Top of Wall or the Bottom of Wall better with a set or existing grade. The "Grade Alignment" feature sets either the Top or Bottom of Wall as the starting point for the Generate Routine. For example, if "Top" is selected, the Panelization routine begins panelizing the Wall by first matching the top block (or Coping if the "Include Coping in Height" is turned on) with the Top of Wall Grade. If your TW grade at Station 1 is set to 100, then the Panelization routine would set the Top of Coping to 100 and build the Wall from the Top –Down. In this case, the Top would be the "Set" or "Existing" grade that would have to be adhered to. Alternatively, if you needed to minimize embedment as much as possible, you could set the Grade Alignment from the

"Bottom" and the bottom block would be set to the minimum embedment elevation, then build the Wall from the Bottom-Up.

As noted above, the normal Panelization routine always starts at the first Station. However, if there was some Datum or grade in the middle of the wall for example that had to be matched (i.e.the TW grade in the middle of the Wall has to hit 110ft), you could select the "Datum" toggle and the Drop Down menu would give you all of the potential TW grades (based on the input Stations) or BW grades to have the Panelization start working from.


Item 8 – Design Tab (April 2017)

8.1 Grid Group Increment

Expanding on the current automatic "Generate" functionality, the Grid Group Increment feature runs an algorithm that attempts to produce a Trial Design for the entire Wall. The algorithm takes into consideration the fact that Geogrid Reinforcement layers generally should be kept at constant elevations (for ease of installation) wherever possible, while still adhering to all Design and Empirical limitations (i.e. Max Crest Height, max reinforcement spacing, etc). Also, the reinforcement lengths should be changed at a reasonable interval along the length of theWall. The Grid Group Increment is the value at which the Reinforcement Length changes from one Group of Panels to the other. For example, if the Grid Group Increment is set at 2.0ft, the Reinforcement Length will step in intervals of 2.0ft from one group of Panels to the next (Note – this is different from the Generation Increment described above, which sets the Trial Design Length for an individual Panel).

 By right clicking on the Elevation portion of the Design Tab and selecting "Select All" or by holding down the SHIFT keyand selecting any number of Panels, you can highlight all the Panels you want Designed. After you have these panels selected, click the "Apply" toggle in the Grid Group Increment area and then input the desired increment that you want the Geogrid to change length along the Wall. For example, if I select all panels, click "Apply", then input 2.0ft , then hit Generate, the algorithm Designs the entire Wall with all layers in horizontal alignment, and the Grid Groups already defined and labelled (Grid Lengths in this example are 5.0ft, 7.0ft, 9.0ft, up to 13.0ft at 2.0ft increments.
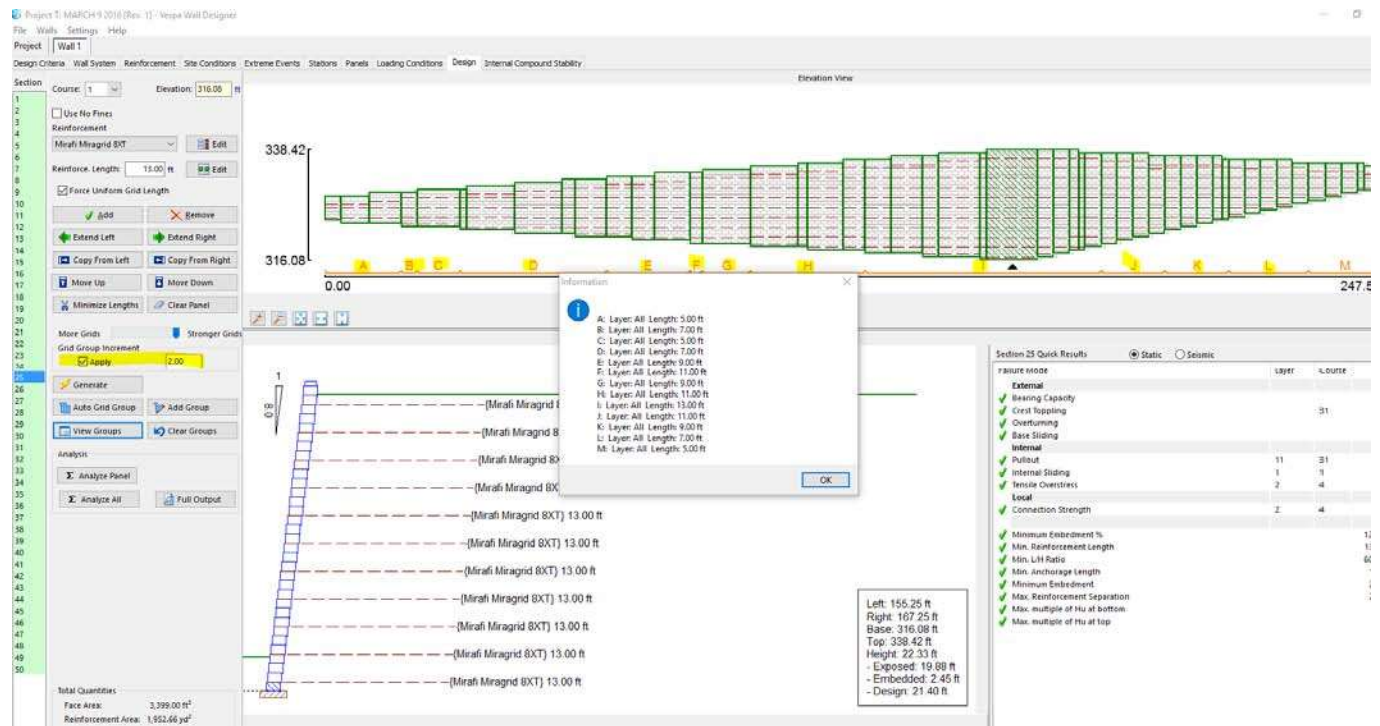
8.2 No Fines Concrete (April 2017)

A practice gaining in popularity is the use of a No-Fines concrete backfill material to create composite gravity Walls. Vespa allows the user to select "Use No Fines" to Generate or define manually a depth of Free Draining Concrete behind the Wall. By selecting "Use No Fines" and hitting "Generate", the algorithm calculates the necessary Wall Depth to resist the standard Conventional Modes of Failure (Base Sliding, Overturning, Bearing Capacity). The user can set the Unit Weight, as this will vary, or the Depth if required.
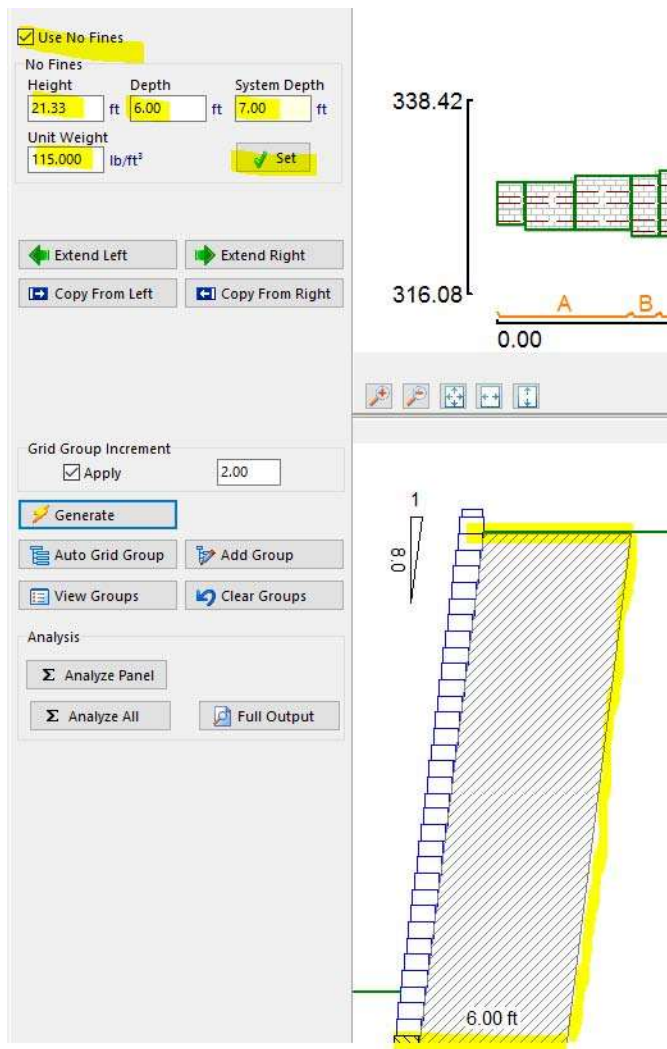
8.3 – Internal Compound Stability (April 2017)

This tab has been moved outside of the Design Tab to provide more space for results to be shown and greater flexibility of input parameters.

Item 9 – File Menu

9.1 Export to AutoCAD

The DXF output has been upgraded to be more user friendly to the CAD Designer. The Facing Units are now drawn as individual blocks with a pline, so the user can move them, manipulate them,etc. Note that these Facing Units are just shown as a representation of how they would appear in the Wall and are not meant to exactly show the position and location of every unit. In some cases, corner units and other specialized units are used which are not shown and which would change the layout of standard units.

Also, if a Wall has a lean or batter, this would distort the actual arrangement of the bond pattern (i.e. bottom of Wall length is actually longer than top of Wall length).
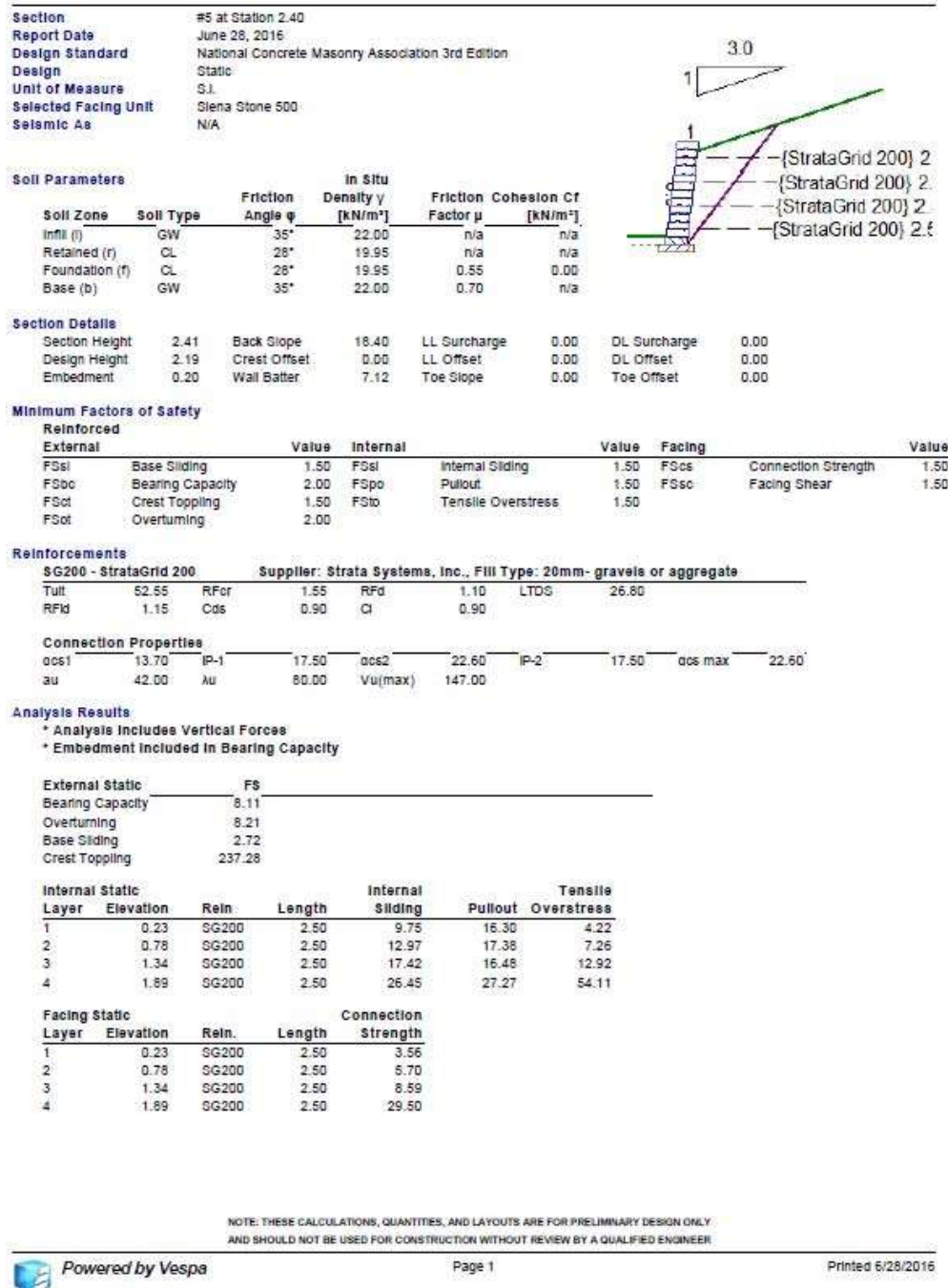
9.2 PDF Output

A number of upgrades have been made to the PDF Report. The Summary Reports, which summarize either the entire Project (Project Materials Summary), or a Single Cross Section (Section Analysis Summary) have been expanded to include more detail and better arrange information.

Section Analysis Summary

The Section Analysis Summary includes all relevant input and output values for the selected Cross Section, which would typically be required by a local Municipal Engineer for review, DOT, etc.

 Project Materials Summary

This Summary Report details all quantities for an entire Project and has been formatted to contain multiple wall systems, includes pins or connectors as required, all infill, drainage, and core fill quantities, as well as reinforcements. Note that the Total Wall Area noted is the sum-total of the facing, coping, and embedded Wall area. The Facing area is the total Wall Area minus the Coping Area.

3.0

— {StrataGrid 200} 2
— {StrataGrid 200} 2.
— {StrataGrid 200} 2.
— {StrataGrid 200} 2.5

Soil Parameters

Soil Zone	Soil Type	Friction Angle φ	In Situ Density γ [kN/m³]	Friction Factor μ	Cohesion Cf [kN/m²]
Infill (i)	GW	35°	22.00	n/a	n/a
Retained (r)	CL	28°	19.95	n/a	n/a
Foundation (f)	CL	28°	19.95	0.55	0.00
Base (b)	GW	35°	22.00	0.70	n/a

Section Details

Section Height	2.41	Back Slope	18.40	LL Surcharge	0.00	DL Surcharge	0.00
Design Height	2.19	Crest Offset	0.00	LL Offset	0.00	DL Offset	0.00
Embedment	0.20	Wall Batter	7.12	Toe Slope	0.00	Toe Offset	0.00

Minimum Factors of Safety
Reinforced

External		Value	Internal		Value	Facing		Value
FSsl	Base Sliding	1.50	FSsi	Internal Sliding	1.50	FScs	Connection Strength	1.50
FSbc	Bearing Capacity	2.00	FSpo	Pullout	1.50	FSsc	Facing Shear	1.50
FSct	Crest Topping	1.50	FSto	Tensile Overstress	1.50			
FSot	Overturning	2.00						

Reinforcements
SG200 - StrataGrid 200   Supplier: Strata Systems, Inc., Fill Type: 20mm- gravels or aggregate

Tult	52.55	RFcr	1.55	RFd	1.10	LTDS	26.80
RFid	1.15	Cds	0.90	Ci	0.90		

Connection Properties

αcs1	13.70	IP-1	17.50	αcs2	22.60	IP-2	17.50	αcs max	22.60
au	42.00	λu	80.00	Vu(max)	147.00				

Analysis Results
* Analysis Includes Vertical Forces
* Embedment Included In Bearing Capacity

External Static	FS
Bearing Capacity	8.11
Overturning	8.21
Base Sliding	2.72
Crest Topping	237.28

Internal Static

Layer	Elevation	Rein	Length	Internal Sliding	Pullout	Tensile Overstress
1	0.23	SG200	2.50	9.75	16.30	4.22
2	0.78	SG200	2.50	12.97	17.38	7.26
3	1.34	SG200	2.50	17.42	16.48	12.92
4	1.89	SG200	2.50	26.45	27.27	54.11

Facing Static

Layer	Elevation	Rein.	Length	Connection Strength
1	0.23	SG200	2.50	3.56
2	0.78	SG200	2.50	5.70
3	1.34	SG200	2.50	8.59
4	1.89	SG200	2.50	29.50

NOTE: THESE CALCULATIONS, QUANTITIES, AND LAYOUTS ARE FOR PRELIMINARY DESIGN ONLY
AND SHOULD NOT BE USED FOR CONSTRUCTION WITHOUT REVIEW BY A QUALIFIED ENGINEER

Item 10 – Walls Menu

The Walls Menu has been enhanced to include a more flexible way to "Copy Wall". In the previous version, beside the "Create Wall" button was an option to "Copy Soil Conditions, Wall System and Reinforcement Type" so that when you Create a new Wall in a Project, these variables are already set and do not need to be entered in again. This option always copied the "First" wall in the Project. Some users wanted the ability to Copy the properties from a different Wall in their Project (i.e the fourth Wall,

for example) when creating the next(fifth) Wall. Our new "Copy Wall" feature in the Walls menu copies the properties from the Current Wall you have selected.

Additional options have been added to Options/Analysis tab.
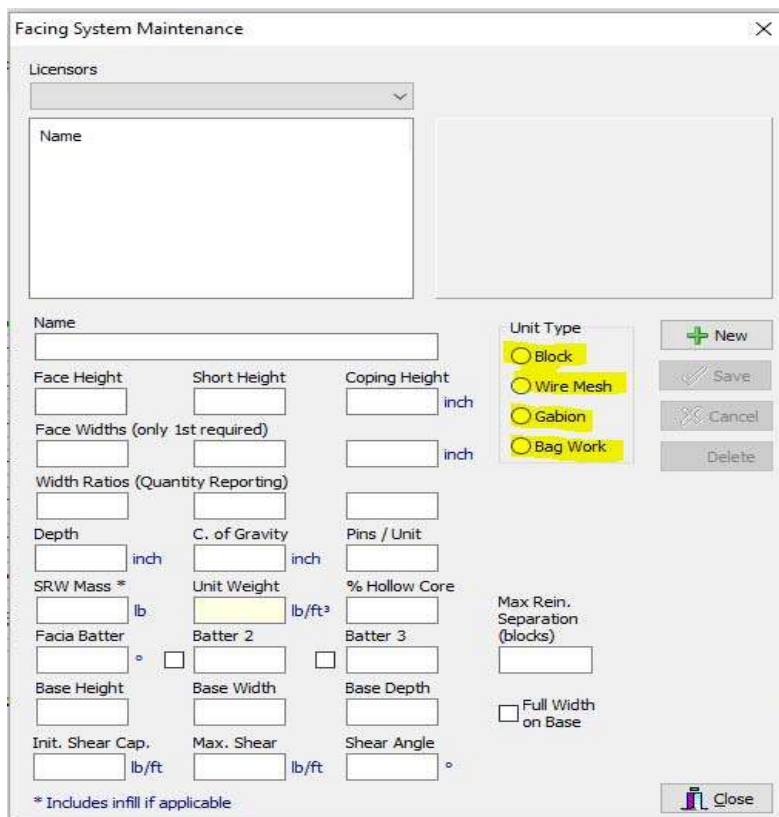
Item 11 – Settings Menu

Vespa now allows the user to create various different Facings in addition to standard"Block" (SRW) type units.

Item 12 – Additional Facing Types

As the Geogrid Reinforcement arrangement can differ depending on the Facing Type, options have been provided to allow the user to detail how the Reinforcement is being configured. For example, if a Wire Mesh type facing is used, the Reinforcement tab changes to provide 4 different options for Reinforcement configuration.

With the Wire Mesh, the user can opt to use "No wrap", Face wrap with secondary wrap only, Face wrap with primary reinforcement and secondary wrap, and Facewrap with primary wrap only. The associated Primary Wrap, Secondary Top Length, and Secondary bottom length input fields are provided as well.

Note that these are provided for Quantity calculation and detailing only, and are not accounted for in the analysis. The chosen methodology is applied, however, for Facings other than SRW Blocks, Facing/Geogrid Connection is not checked.

Item 13 NCMA Design Methodology

(April 2017)

A change has been made to the way Vespa handles the vertical component of the Dead Load forces for use as a "Resisting Force". Per Section 7.4.1.2 of the NCMA DMFSRW 3rd Edition, the vertical component of the exterior Dead Load is to be neglected as a resisting force if the Dead Load offset is outside of the internal influence distance (dint) or the reinforced zone, defined by Lbeta. In Vespa, As the Dead Load was still being applied to the exterior of the Reinforced Mass as a destabilizing force, the vertical component of that load was also still included as a stabilizing force (Pqdv). Although accurate, this was a deviation from the NCMA recommendation so the Pqdv has been removed as a stabilizing force if outside of the reinforced zone, despite the fact that these vertical forces do exist. This is a conservative change and only applies when the "include vertical forces" is turned on in the Settings/Options.

Item 14 DXF Updates

In order to make the DXF output more "print ready" and useful in the field, we have added a few upgrades.

We have added 3 rows under the Wall that show the BOW grade, Min. Applied Bearing Capacity, and Reinforcement Requirements for each Panel.
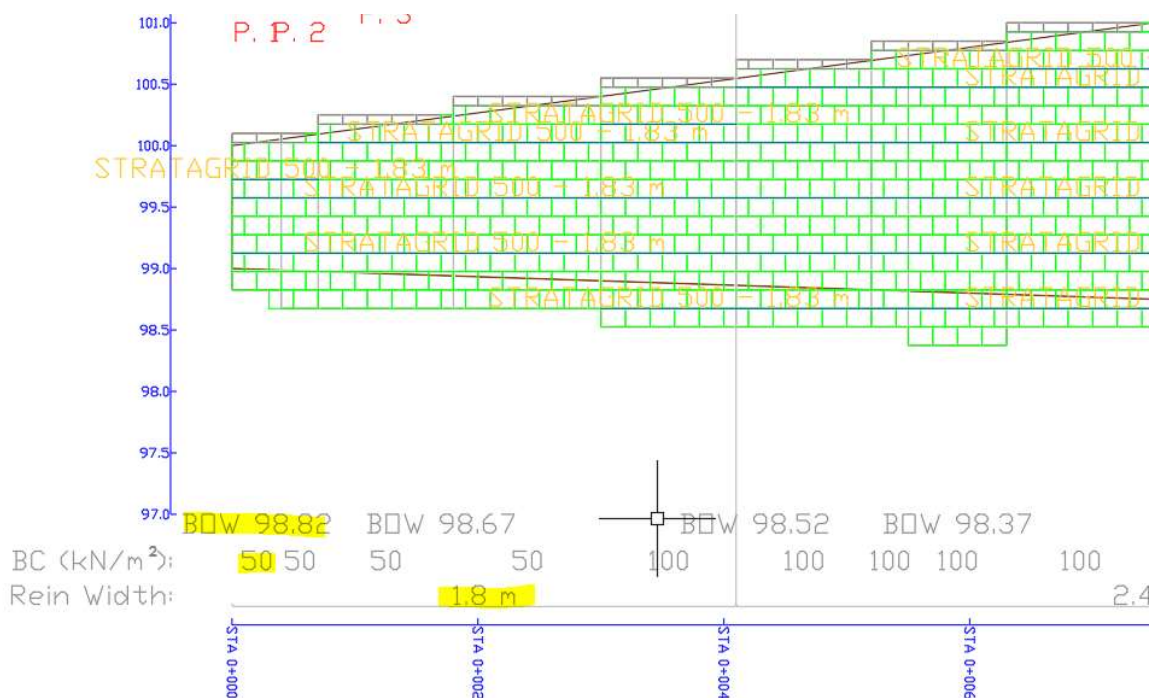
We have also added the Station Data (Station and TOW Grade) to the elevation view so the user can reference the original Grades Input. As well, these have been added to another table in the DXF, along with the Crest/Toe inputs.
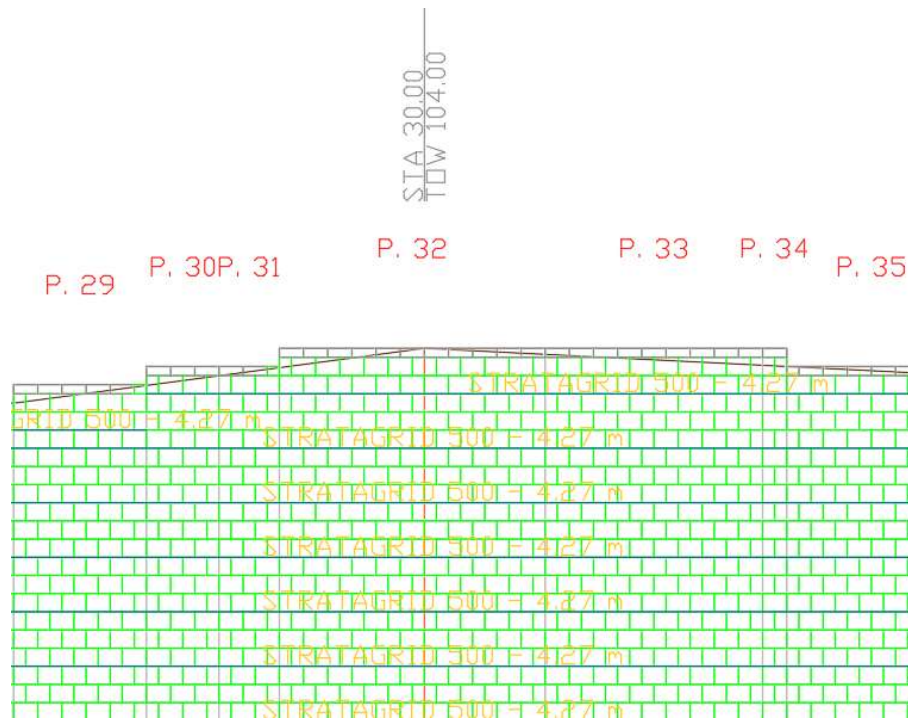
All of these elements are on independent layers so they can be easily turned off if the user does not want them.

Item 15 – Bug Fix (April 2017)

In the last (April) build, for NCMA Methodology, for the calculation of applied Bearing Load the Dead Load was being used in place of the Live Load. Therefore, for very large Dead Loads, the Dead Load would be essentially applied twice and results would be conservative for applied bearing load. This has been fixed.

Item 16 – April 2017

For AASHTO 2010 and 2015, for calculation of Bearing Capacity (Bearing Resistance), we had originally adopted the convention of using "L" as the width of the Wall, as opposed to B (which includes the facing). The AASHTO direction on this was that either L or B could be used, depending on the thickness of the face (i.e. if the face was "thick", then it should be B). As this is a vague term and likely relative to the MSE type walls, we erred on the conservative side and used "L", thereby ignoring the facing. The consensus of users is that B is more widely accepted as most block facings are at least 1ft thick or larger and there is also a base to consider. This change has been made.

Item 17 – April 2017

Previously, in AASHTO 2010 and 2015, the "Overturning" check is a comparison of the calculated base eccentricity to the maximum allowable eccentricity. A ratio of e/emax was used to express the fact that the emax was not being exceeded. This check was the only method AASHTO provided for looking at overturning. We have expanded the Overturning check to be more in line with a classic overturning analysis, with Mresisting/Moverturning being the actual Overturning check. We have moved the current eccentricity check to the "empirical checks" portion of the analysis, allowing the user to specify the max eccentricity in the Design Criteria.

Item 18 (Bug Fix). (April 2017)

For AASHTO 2010, the Pae, Paell, and Paedl were being calculated using the full Kae value as they are in 2015. In 2010, however, the Static loads are still being applied, so a reduced value for Kae (reduced for the static portion) should be used and is now changed. In short, the previous build overestimated applied Seismic loads and was therefore overly conservative.

Item 19 (May 2017)

Two items were identified which, when changed, no longer wipe out the existing Design. Vespa allows you to change the Crest Offset and Crest Elevation without causing a repanelization and/or wipe out the Geogrid Design that is in there. The other variable is the Block. If a Block that has the identical face height and width is substituted for the block in the current Design, the Panelization will not occur again.

Item 20 (May 2017)

NCMA, Conventional, Seismic, Bearing Capacity

On page 129 of the NCMA Design Manual, Third Edition, the following equation is given for the Seismic Eccentricity, eseismic.

9.7.1.3: Bearing Pressure (Refer to Section 6.4.5)

pplied bearing stress at the base of the leveling pad is calculated as:

$$Q_{aseismic} = \frac{[W_{gr} + P_{sV} + P_{qV} + 0.5\Delta P_{dynV}]}{B'_c} \qquad \text{[Eq. 9-42]}$$

$$B'_a = W_u + h_{lp} - 2e_{seismic} \qquad \text{[Eq. 9-43]}$$

$$e_{seismic} = \frac{\left[k_{hint}W_u\left(\frac{H}{2}\right) + P_{sH}\left(\frac{H}{3}\right) + P_{qdH}\left(\frac{H}{2}\right) + 0.5\Delta P_{dynH}\left(\frac{H}{2}\right)\right] - W_w X_w}{W_u} \qquad \text{[Eq. 9-44]}$$

The moment arm shown for the resisting moment is Xw, which is the distance from the toe of the wall to the center of gravity. This is contrary to the Conventional Static version of the same equation, where the moment arm is correctly shown as ew, which is the moment arm from the center of the block. Vespa followed equation 9-44 due to this misprint, but has been changed to reflect the ew requirement shown in Static.

Item 21 (Bug Fix) (May 2017)

NCMA, Reinforced

In the case where a Dead Load was applied to the top of the wall, and the Dead Load Offset was beyond the Reinforcement Zone (DLoffset>L), an error in logic was causing the Dead Load to still be included as part of the Resistance for some of the calculations. Specifically, the contributing effect was being applied to the Overturning Resistance (which would overestimate the overturning resistance) and the applied Bearing Capacity (which would produce a conservative result) in the Static calculations. It did not affect the Base Sliding Calculation, which is typically the critical mode of failure in External Stability analysis. In Seismic, the Dead Load contribution was being applied to Overturning, Sliding, and Bearing Capacity. The error only occurred when the Dead Load Offset was outside of the Reinforced Zone and inside of the influence zone (L+H+h).

Item 22 AASHTO -Inclusion of Wall Batter (August 2017)

AASHTO 2010 and 2015 state that for a Wall Batter less than 10 degrees, the Wall is assumed to be vertical. Based on discussions with Designers, we have concluded that while we can apply this principle to the calculation of Ka (and Kae), the model is more accurate if we include the Wall Batter in the Geometry when calculating Wall weights, moment arms, etc. As a result, we will now use the actual wall batter, even if it is under 10 degrees, to more accurately model the Walls with respect to Geometry as shown in the figure below. The applied external forces will still act on the vertical interface. Note that the weight W4 is not included as it lies outside of the vertical interface. This is not consistent with the NCMA approach, which simply approximates Wri as a rectangular shape with height (H) and depth (L). Not including W4 may therefore give an overly conservative result. To check this, you can set your Wall batter to zero and compare.
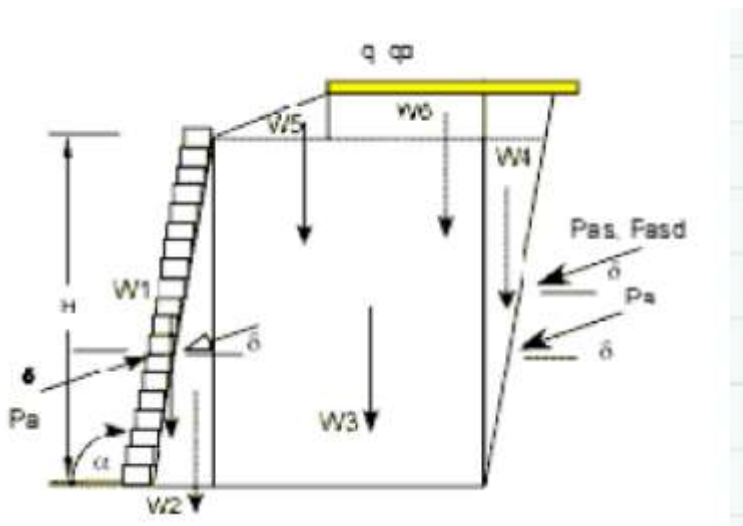
Item 23 AASHTO 2015, Seismic (August 2017)

One of the main changes to AASHTO for 2015 was the calculation of total Horizontal Thrust (Seismic). Thf had to be checked for three conditions to get the maximum value. The three conditions (per C11.6.5.1) included: 1) 100%Pae+50%Pir, 2) 50% Pae +Pir, and 3) Unfactored Static Loads+Pir. This was incorporated when 2015 was included in Vespa. The max of these was used where dictated by AASHTO

(Base Sliding Calculation). Upon reviewing this again recently, it was determined that if the intent was to always look at these combinations, they should also be applied to the new Overturning Moment calculation that was added in April of 2017 (on top of the existing eccentricity check that was already in place). As such, the load combinations are used to determine the maximum overturning moment and bearing eccentricity. This is not explicitly defined in AASHTO, however, we believe it is in line with the intent of C11.6.5.1.

Item 24 – AASHTO Load Factors (August 2017)

When AASHTO 2015 was introduced to Vespa, the Load Factors/Resistance Factors were separated from being a single variable that was either set to a Min or Max value (depending on the use), to what we have now, which is a separate variable for the Minimum Load/Resistance Factor and another one for the Maximum Load/Resistance Factor. This made the coding clearer and also allowed us to format the Design Criteria table better. There was one case reported where upon updating the database, a duplicate value for the Live Load Factor (Max.) was created and it was a corrupt value (set to 0 instead of 1.75). Additional checks have been added to ensure duplicate values are not created, however, for older files, ensure to check Load/Resistance factors are not duplicated or have incorrect values.

Item 25 – Changing Blocks/Geogrids on Existing Designs (August 2017)

Some users wanted the ability to change the SRW units/Facing in a Wall, without wiping out their Geogrid Reinforcement Design. We have now added the ability to do this. If you select a new Facing, different from the current one being used in the Design, as long as it has the same face dimensions (Block Height and Block Length) AND it has connection testing with the existing Geogrids, the Facing will be replaced. Keep in mind, this does not automatically mean it will pass all criteria the previous facing may have (as connection properties may change), so you have to run the analyze all routine again.

Updates – September 2017-December 2017

Item 26. Significant performance improvement when generating or analyzing, or copying left/right

Item 27. Apply All added to Loadings screen. This makes it quicker to apply loading to all panels instead of extend left and right.
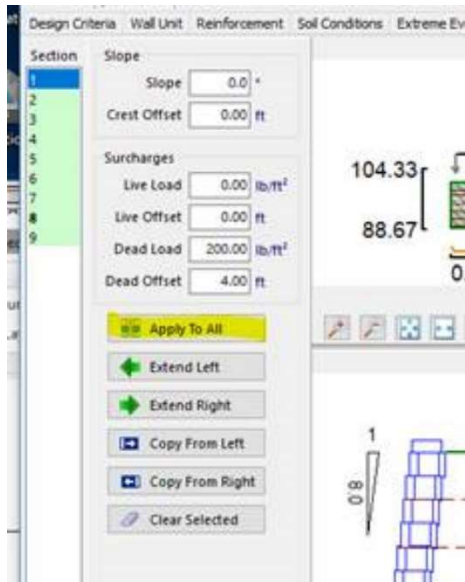
Item 28. Panelization improvement – In some situations, previous panelization routine could create ½ block width wider than it needed to.

Item 29. Change to never consider coping in calculating needed embedment (now includes in % of height).

Item 30. Improved handling of RsTop/Bottom during generation, and in conjunction with a block's max. separation

Item 31. Better handling of potential pullout errors on a grid too close to grade

Item 32. Improved handling of uniform grid lengths

Item 33. Soil type on reinforcement screen now applies to all selected reinforcements, no longer allows diff. settings for each

Item 34. Option to ignore crest toppling during generation. Sometimes this is useful if you know you will be putting a small secondary layer of grid at the top, but do not want to have your top layer of grid 1ft from the top (and get crazy lengths).
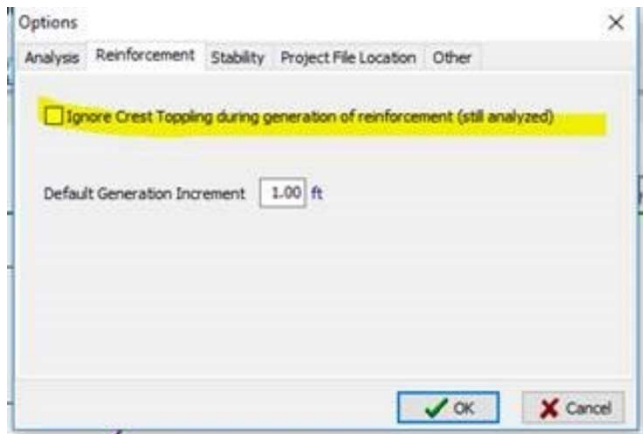
Item 35. AU (Australian) Code: structure importance classification now on project screen and better PDF output.

Item 36. Can edit file names for DXF export

Item 37. Improvement on how NCMA and AASHTO handle load offsets.   For External Stability, AASHTO and NCMA do not really provide a graduated way of reducing surcharge loads if they are offset beyond the back of the reinforced zone. We have implemented a simple approach where the load would be reduced according to the offset distance. For example, in NCMA, a factor (called a qdfactor) would be applied to the Live/Dead load according to the following:

$Pq = q \cdot Hext \cdot Kaext \cdot qfactor$

where qfactor equals:

IF $qoffset \leq Lb$ then $qfactor = 1.00$

IF $qoffset > (Lb + Hext)$ then $qfactor = 0.00$

Else $qfactor = [(Lb + Hext - qoffset] / Hext$

For AASHTO it would be:

$Pq = q \cdot Hext \cdot Kaext \cdot qfactor$

where qfactor equals:

IF $qoffset \leq Lb$ then $qfactor = 1.00$

IF $qoffset > 2H$ then $qfactor = 0.00$

Else $qfactor = (2H - qoffset) / (2H - Lb)$

Item 38. NCMA Seismic Analysis

For situations where the values entered for phi angle, back slope and Seismic Kh are incompatible with the M-O analysis, Vespa previously brought up the warning (which the user had to agree to) which stated that an "Alternate Seismic Method" was used. This Alternate Seismic method was the "Trial Wedge" approach, which analyzed potential failure planes behind the reinforced zone on 1 degree increments to solve for the maximum thrust (more information available upon request). In the latest version of Vespa, we are requiring that the user more explicitly agree to this approach by selecting it as an option in the Settings/Options/Analysis. When the option is turned "ON", the Trial Wedge Approach is automatically run when the condition exists:

$$\phi_r - \beta_{ext} - \Theta_{ext} < 0$$

If the option is not on, the industry standard "stop-gap" method of setting this term = 0 when it is negative.

January 2018

Item 39. Offset Loads and Reduction Factors - Option

As noted in Item 37, we have provided a way to reduce Live and Dead loads based on Offset Distances.

However, we have updated Vespa to have this as an Option the user selects for NCMA, rather than just automatically utilizing it, as this is not explicitly discussed in the NCMA Design Methodology.

As well, we have included a further method to reduce loads for internal stability based on offset as described below.

To review, for AASHTO, External Stability, a Factor for the Live and Dead load is calculated as follows:

$$q_{lFACTOR} := \begin{Vmatrix} \text{if } q_{loffset} \leq L_\beta \\ \quad \begin{Vmatrix} 1 \end{Vmatrix} \\ \text{else if } q_{loffset} \geq 2 \cdot H \\ \quad \begin{Vmatrix} 0 \end{Vmatrix} \\ \text{else} \\ \quad \begin{Vmatrix} \dfrac{(2 \cdot H - q_{loffset})}{2 \cdot H - L_\beta} \end{Vmatrix} \end{Vmatrix} = 1$$

$$q_{dFACTOR} := \begin{Vmatrix} \text{if } q_{doffset} \leq L_\beta \\ \quad \begin{Vmatrix} 1 \end{Vmatrix} \\ \text{else if } q_{doffset} \geq 2 \cdot H \\ \quad \begin{Vmatrix} 0 \end{Vmatrix} \\ \text{else} \\ \quad \begin{Vmatrix} \dfrac{(2 \cdot H - q_{doffset})}{2 \cdot H - L_\beta} \end{Vmatrix} \end{Vmatrix} = 1$$

This factor is then applied to the Driving Live and Dead forces, F2 and F3. Because AASHTO is silent on how the offset load can be gradually reduced, we have made the conservative assumption that the maximum influence distance from the back of the Wall is equal to 2H. This factor is automatically utilized during the analysis.

For AASHTO Internal Stability, the result is not graduated and basically applies the full Live or Dead Load if the Offset is within the Reinforced Zone.

$$q_{lAPP} := \begin{Vmatrix} \text{if } q_{loffset} > L \wedge q_{loffset} > d_{int} \\ \quad \begin{Vmatrix} 0 \; \dfrac{lbf}{ft^2} \end{Vmatrix} \\ \text{else} \\ \quad \begin{Vmatrix} q_l \end{Vmatrix} \end{Vmatrix} = 0 \; \dfrac{lbf}{ft^2}$$

For NCMA, the user is given the option to either run "Standard" NCMA handling of offset loads as described in the NCMA Manual (3rd Edition), OR, choose to more accurately reflect the load offsets according to the following calculations. The Option to choose this is in the Settings:

For External Stability, the Qdfactor and Qlfactor are calculated as follows:

$$Q_{dfactor} := \text{if } q_{doffset} \leq L_\beta \quad = 0.848$$
$$\left\| 1 \right.$$
$$\text{else if } q_{doffset} > L_\beta + H_{ext}$$
$$\left\| 0 \right.$$
$$\text{else}$$
$$\left\| \left(\frac{L_\beta + H_{ext} - q_{doffset}}{H_{ext}} \right) \right.$$

$$Q_{lfactor} := \text{if } q_{loffset} \leq L_\beta \quad = 0.848$$
$$\left\| 1 \right.$$
$$\text{else if } q_{loffset} > L_\beta + H_{ext}$$
$$\left\| 0 \right.$$
$$\text{else}$$
$$\left\| \left(\frac{L_\beta + H_{ext} - q_{loffset}}{H_{ext}} \right) \right.$$

For Internal Stability, a similar approach to the NCMA method is used, however, instead of the 2V:1H influence line assumed by NCMA, a more conservative 1H:1V influence line is used. However, although we are more conservative from the distribution point of view, we then apply this reduction to all internal modes of failure (Tensile Overstress, Pullout, Connection), as opposed to the NCMA, which just applies it to Connection.

As with NCMA, the intercept height on the back of the Wall facing is calculated (assuming a 1H:1V distribution.

Live Load

$$E_{qLinfl2} := \text{if } q_{loffset} \geq q_{offsetmax} \quad = 5.1 \, ft$$
$$\left\| 0 \, ft \right.$$
$$\text{else}$$
$$\left\| H - \left(\frac{q_{loffset} - h_s}{1 - \tan(\omega)} \right) \right.$$

Dead Load

$$E_{qDinfl2} := \text{if } q_{doffset} \geq q_{offsetmax} \quad = 5.1 \, ft$$
$$\left\| 0 \, ft \right.$$
$$\text{else}$$
$$\left\| H - \left(\frac{q_{loffset} - h_s}{1 - \tan(\omega)} \right) \right.$$

An individual Load Reduction Factor is then calculated for each layer of Geogrid as follows:

$$qlf_n := \begin{cases} \text{if } E_{qLinft2} = 0 \ ft \\ \quad \| 0 \\ \text{else if } n = 1 \\ \quad \| \text{if } E_{qLinft2} \geq \dfrac{\left(E_{n+1} + E_n\right)}{2} \\ \quad\quad \| 1 \\ \quad \| \text{else} \\ \quad\quad \| \dfrac{E_{qLinft2}}{\dfrac{\left(E_{n+1} + E_n\right)}{2}} \\ \text{else if } n = Nl \\ \quad \| \text{if } E_{qLinft2} \geq H \\ \quad\quad \| 1 \\ \quad \| \text{else} \\ \quad\quad \| \max\left(0, \dfrac{E_{qLinft2} - \dfrac{\left(E_n + E_{n-1}\right)}{2}}{H - \dfrac{\left(E_n + E_{n-1}\right)}{2}}\right) \\ \text{else} \\ \quad \| \text{if } E_{qLinft2} \geq \dfrac{\left(E_n + E_{n+1}\right)}{2} \\ \quad\quad \| 1 \\ \quad \| \text{else} \\ \quad\quad \| \max\left(0, \dfrac{E_{qLinft2} - \dfrac{\left(E_n + E_{n-1}\right)}{2}}{\dfrac{\left(E_n + E_{n+1}\right)}{2}}\right) \end{cases}$$

Bottom Layer (n+1 = 2, n = 1)

Top Layer

←

Intermediate Layer

Item 40 – New PDF Output Options

In the PDF output dialogue box, you can now select multiple Panels from any Wall within a given Project. For example, if you wanted to print out the detailed calculations for Panel 5 in Wall 1, Panel 3 in Wall 2, and Panel 10 in Wall 3, you can select each of these Panels in the new selection box and they will be compiled in a single report.
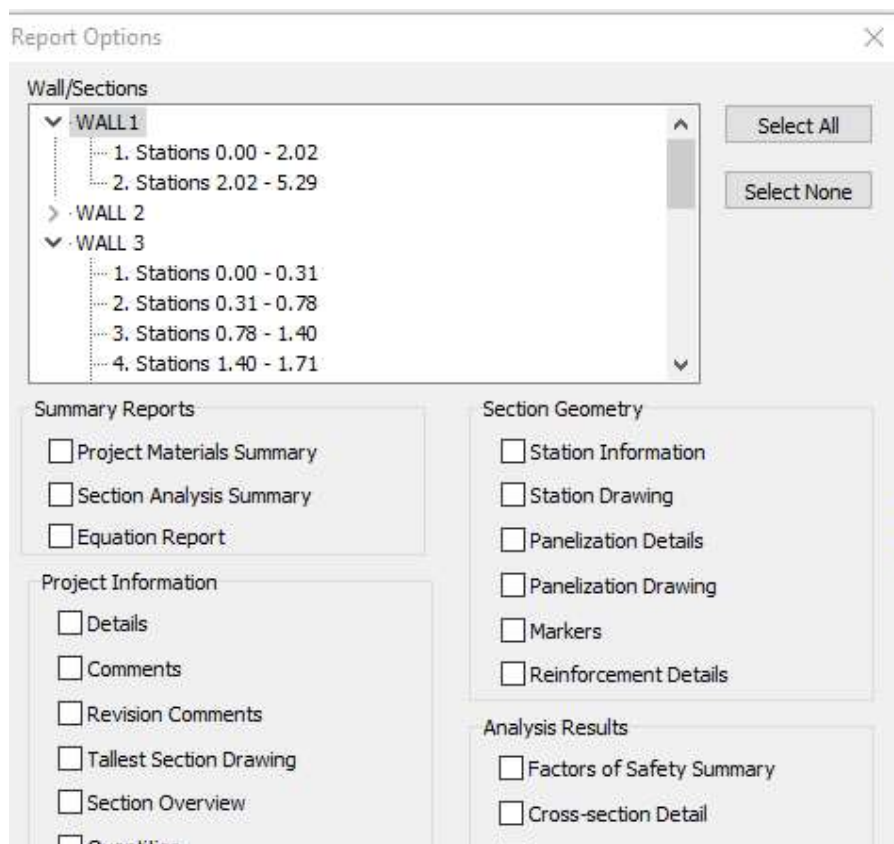
Item 41 – March 2018 - Improvement when Switching between Methodologies

In a given Project, if the user was switching between different Design Methodologies, such as NCMA to AASHTO, the user would have been required to go to the Reinforcement Tab to reset the related Connection Data. New coding has been added to ensure the user can move directly to the Design/Analysis without first going to the Reinforcement Tab to get the AASHTO Connection Creep and Durability values.

Item 42 – March 2018 – NCMA Crest Toppling

Some Vespa users noted that the NCMA Seismic Crest Toppling calculation yielded a different result than the "same" Overturning calculation when analyzing an independent conventional Wall of the same height and loading conditions. This is due to some inconsistencies in the NCMA Design Methodology. For Static Crest Toppling, NCMA currently uses Beta(int) and Ka(int) in the analysis. Calculation of Beta(int) is based on the full Wall height, but should be based on only the smaller Crest (gravity) height of the Wall. As such to be more accurate, we need to create a couple of new variables to account for this.

First, for Static Crest Toppling, we create a new variable called hmaxCREST

$$h_{maxCREST} := H_{unreinf} \cdot \tan(\beta)$$

This variable is then used to calculate a new Beta value (just for Crest Toppling)

$$\beta_{crest} := \left\| \begin{array}{l} \text{if } h_s < h_{maxCREST} \wedge \beta_{offset} > 0 \text{ ft} \\ \quad \left\| \operatorname{atan}\left(\dfrac{h_s}{H_{unreinf}}\right) \right. \\ \text{else} \\ \quad \left\| \beta \right. \end{array} \right\| = 18.4 \text{ deg}$$

Based on the new variables, a new Kaint(CREST) is calculated. Note that we are still using the infill friction angle instead of the Retained friction angle as it is assumed the "reinforced" material extends to close to the top of the Wall.

$$Ka_{intCREST} := \frac{\cos(\phi_i + \omega) \cdot \cos(\phi_i + \omega)}{\cos(\omega) \cdot \cos(\omega) \cdot \cos(\omega - \delta_i) \cdot \left(1 + \sqrt{\dfrac{\sin(\phi_i + \delta_i) \cdot \sin(\phi_i - \beta_{crest})}{\cos(\omega - \delta_i) \cdot \cos(\omega + \beta_{crest})}}\right)^2} = 0.3$$

We assess the effect of offsets on applied Live and Dead Loads as follows (if this option is ON)

$$Q_{lfactorCREST} := \left\| \begin{array}{l} \text{if } q_{loffset} > H_{unreinf} \\ \quad \left\| 0 \right. \\ \text{else} \\ \quad \left\| \left(\dfrac{H_{unreinf} - q_{loffset}}{H_{unreinf}}\right) \right. \end{array} \right\| = 0 \qquad Q_{dfactorCREST} := \left\| \begin{array}{l} \text{if } q_{doffset} > H_{unreinf} \\ \quad \left\| 0 \right. \\ \text{else} \\ \quad \left\| \left(\dfrac{H_{unreinf} - q_{doffset}}{H_{unreinf}}\right) \right. \end{array} \right\| = 0$$

Crest Toppling calculations continue per NCMA based on the above new variables.

For Seismic Crest Toppling, the major difference here is the assumption that the Internal Inertial Angle should be applied to the Kae (int) equation. However, if this small (Crest) Wall was considered as an

independent gravity wall, the External Inertial Angle would apply, which we allow deflection to apply. As a result, the current Seismic Crest toppling often results in higher seismic loads than the equivalent Gravity Wall. To balance this, we have included the Option in Settings to allow the khext to be used in the Crest Toppling analysis.
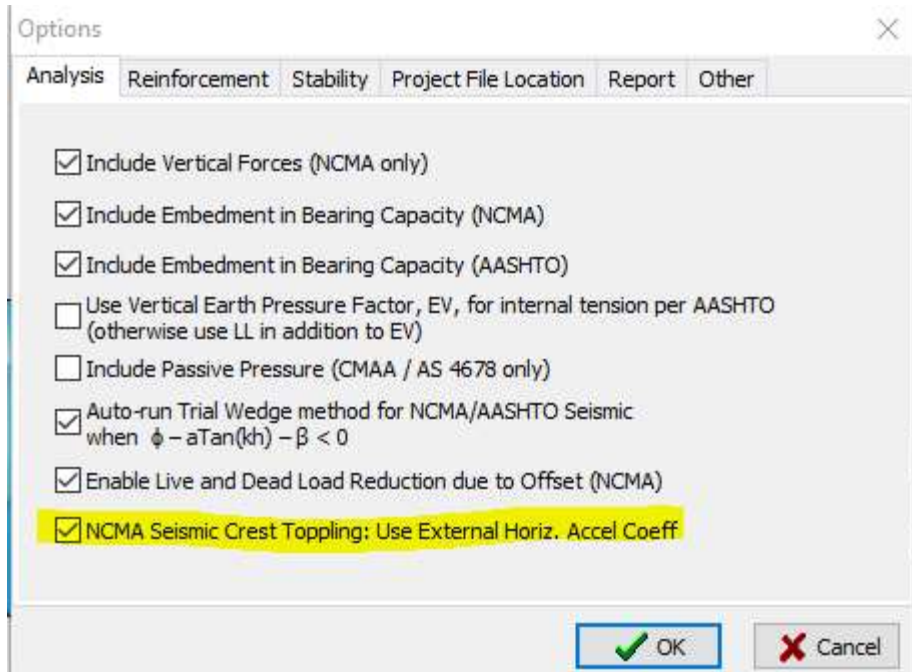
The new Kae(int) CREST would then be:

$$K_{AE.intCRESText} := \frac{\cos\left(\phi_i + \omega - \Theta_{ext}\right) \cdot \cos\left(\phi_i + \omega - \Theta_{ext}\right)}{\cos\left(\Theta_{ext}\right) \cdot \cos\left(\omega\right) \cdot \cos\left(\omega\right) \cdot \cos\left(\delta_i - \omega + \Theta_{ext}\right)} \cdot \frac{1}{\left(1 + \sqrt{\dfrac{\sin\left(\phi_i + \delta_i\right) \cdot \sin\left(\phi_i - \beta_{crest} - \Theta_{ext}\right)}{\cos\left(\delta_i - \omega + \Theta_{ext}\right) \cdot \cos\left(\omega + \beta_{crest}\right)}}\right)^2} = 0.3$$

 (Note – because the new approach does not strictly follow NCMA, although it is more correct, the user will have to select the Option in Settings).

Item 43 – Reformat PDF Output

The detailed calculation output tables have been reformatted to be more consistent from Methodology to Methodology, particularly for gravity structures, highlighting the applied loads, then resisting loads, followed by the associated FS or CDR value.

Item 44 – NCMA : Addition of Qd and Ql Factor for Internal Sliding (May 2018)

As noted in Item 39 above, we provided the user a way of accounting for offset Live and Dead Loads in a "graduated" way. By applying a Qd (Dead Loads) and Ql (Live Loads) Factor to External Stability, we are able to gradually reduce loads with offsets. Until this build, we only applied these factors to External Stability. We are now applying these factors to the Internal Sliding Calculations as well. As the actual Wall height decreases with each Geogrid Elevation analyzed, application of the Ql and Qd Factors from External Stability will be a conservative approach.

Items from this point forward relate to Vespa and Vespa 2

Item 45 – NCMA: No Fines Analysis

The current No Fines Analysis follows the typical NCMA Conventional Analysis, where the Base Sliding Resistance is calculated based on the friction along the Gravel Base. Some users have indicated that while the Base Block (SRW) may be on the Gravel Base, the "No Fines" portion of the mass may not be.

As such, part of the Mass is sliding along the Gravel Base, and part is bearing directly on the Foundation Soil. To err on the conservative side, we have changed the standard analysis to utilize the Foundation Soil Friction Angle in the Base Sliding Resistance Calculation. If this is not the case in your application (entire mass is bearing on a Gravel Base), increase the Foundation Soil Friction Angle for your base sliding calculation. Make sure to change it back to check Bearing Capacity calculations.

Item 46 – Formatting Bug in PDF output

For Reinforced Walls, the FS/CDR Table in the PDF output was showing the values for Internal Stability for each layer of geogrid in reverse order relative to their elevation. This has been fixed. This was just a formatting issue – the values are correct, they were not being displayed in the correct order.

Item 47 – Explanation of Crest and Toe Geometry Extrapolation

Some users have asked for a more comprehensive explanation of how Crest and Toe Geometry are extrapolated between Stations and Panels. For this build, we have modified the way it interpolates between Stations that we feel is more intuitive.

When entering stations, crest and toe information can optionally be entered for each station. When the crest or toe option is first turned on, the corresponding fields will show in red as "calc" indicating that, if nothing is entered for any of these they will be calculated for you once you complete your entries and either press the Validate button or move off the Stations screen.

Along with entering offsets, an additional option allows entry of either elevations (feet or meters) or slopes (degrees). This option can be toggled at any time, entered values will be converted as needed. In order to be of any purpose at least one offset and one elevation/slope must be entered for crests and the same for toes (a single offset could be entered for one station and a single elevation/slope entered in a different station).

Station Resolution
When validation of stations occurs, entered values are extrapolated in order to fill any missing ('calc') values. If nothing is entered for a value in the first or last stations these will default to minimum values – offsets will be set to zero and elevation will be set to that station's Top value for crests or Bottom value for toes (will show as 0 degrees if entering as slopes).

For all other missing values, they are calculated based on that station's X position along a plot line between the containing stations having values with Origin as the X value and the offset or elevation as the Y values (note that even if entered as slopes, this process uses the corresponding elevation values for this process). First missing offsets are calculated, and then elevations.

A simple example to understand would be a case of 4 stations entered at equal intervals with the first crest offset set to 0 and the last set to 12 feet. This process would fill in the two missing offsets with 4 and 8 feet. Likewise, if crest elevations were entered for first and last stations of 10 and 13 feet, the missing elevations would be set to 11 and 12 feet. If slopes are being displayed instead of elevations, these will then be calculated accordingly.

Panel Resolution
When the stations are "panelized" using the selected wall unit, crest and toe data entered and/or calculated for stations is applied to each of the generated panels. For each panel, the set of stations affecting it are determined. The offsets and elevations for these are then plotted and from that the points on these lines for the left and right sides of each panel can be determined. In addition to the left and right sides of the panels, there may be one or more stations within the panel's bounds. The Design Height for all of these is computed and the largest Design Height determines the controlling values for this station. For example, if each panel in a wall is growing in Design Height from left to right then the right side of each panel will be the controlling station and values at this position will be used to set the crest and toe values for that panel.

Item 48 – NCMA Load Offset Option (May 2018)

In Item 39 above, we explained an option that was added in 2018 to allow the user to conservatively reduce Live and Dead Loads based on offsets. This option expands on what the NCMA methodology current offers.

We have made two changes to how this option works, based on feedback from users.

1. For Internal Stability Analysis, the NCMA provides guidance for load reductions due to offsets for Connection and Crest Toppling. Since this method was already in place, we maintained this approach for analyzing Connection loads, even when the "Load Reduction" option was ON. We applied the new offset reduction method to Tensile Overstress and Pullout. This was inconsistent due to the fact that the NCMA approach assumes a 2V:1H line of influence for Connection, and we were applying a 1H:1V line of influence for Tensile and Pullout. Therefore, if the Load Reduction option is "ON", the Geogrid Loads (Fgn) are reduced the same for all modes of failure (according to the method described above in Item 39). If the option is OFF, the Connection loads are reduced according to the NCMA.

2. For Internal Sliding, we were conservatively not applying the load reduction factors that we did for External Stability (Base Sliding). We have included these to be more consistent when the Load Reduction option is ON.

Item 49 – NCMA Seismic

For the calculation of the Weight of Slope above the Wall (Wbeta), Vespa uses the unit weight of the Retained Soil, as experience has shown that typically retained or native soils compose the slope as opposed to the infill soil. Recent review of our code indicated that for the calculation of the modified slope weight (W'beta), the infill unit weight was being applied. We have changed this to be the retained unit weight to be consistent. The difference in results will likely be negligible in most cases.

Item 50 – PDF Output

To make it easier for users to identify which Cross Sections they may want to print out (Section Analysis Summary, Detailed Calculations, Equation Report), the Grid Group Designation (A,B, C, etc) is now shown beside the Panel in the PDF output dialogue box. As shown in the image below, the Panels show an orange letter beside them indicating which Grid Group they are part of. Note that the Highest Panel in each Grid Group is shown with an asterisk beside the letter. Using this convention, a user can easily select the highest panel of each grid group to print Section related data.

Item 51 – NCMA Crest Toppling

In Item 42 (refer above), we discussed how we dealt with slopes above the Wall Crest. As noted, the NCMA Methodology uses Beta(int) that in the Ka equation for Crest Toppling. Beta(int) is calculated for all internal stability calculations based on the entire Wall Height (equivalent slope when you have broken back conditions is based on 2H). We have identified that as not being completely accurate, as we are just looking at the Crest Height in the Crest Toppling Calc. As such, we revised the Beta (int) specifically for Crest Toppling, which yielded a new Ka(int)CT. While this approach is more correct and more accurately reflects/models the slope conditions, it is technically not NCMA compliant. As such, we have added another option in the Settings/Options/Analysis menu which allows the user to use the Alternate Ka(int) for Crest Toppling as shown below.
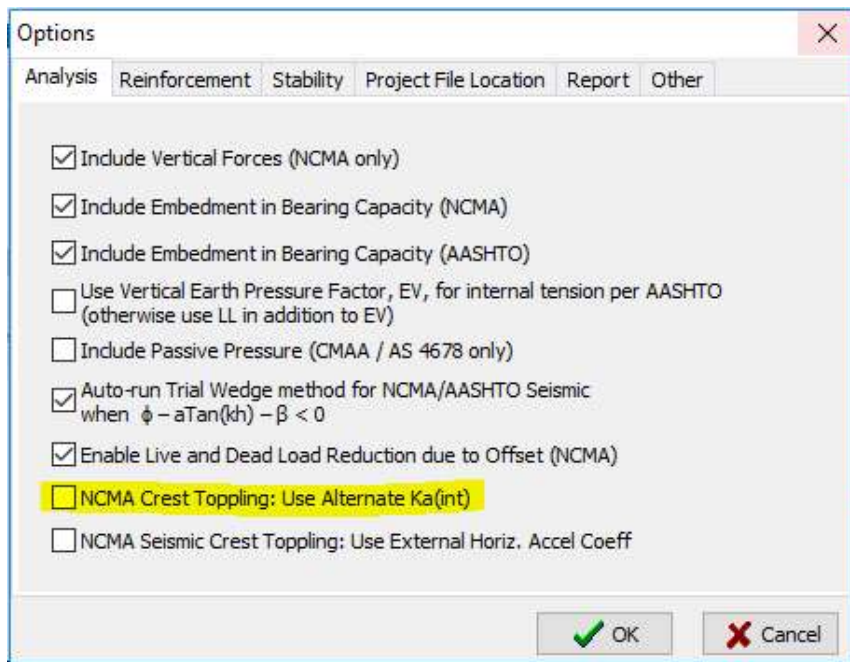
Item 52 – Additional Information in "Project Information/Quantities"

In the Project Summary, we have added the Tallest Panel Height and Longest Reinforcement Length, as these are typically good indicators of the size/scope of the Wall.

Project Summary

Quantities

Wall Length	102.36 ft
Total Wall Area	818 ft²
Cap Area	62 ft²
Exposed Area (includes cap)	736 ft²
Embedded Area	82 ft²
Tallest Panel Height	11.53 ft
Longest reinforcement length	12.00 ft
Base soil volume	5 yd³
Infill soil volume ‡	44 yd³

Reinforcement

SG200 - StrataGrid 200	116 yd²

Item 52 – Split/Merge Panels and Loading – August 2018

When a Panel is Split, the loading from the original Panel is automatically copied to the new (Split) Panel. The Slope and Slope Offset was being automatically copied, but Live or Dead loads were not automatically being copied over. This has been corrected.

Item 52 – Top of Base Label is DXF – August 2018

A bug was fixed where the TOB (Top of Base) value in the DXF was not being shown in the last panel.

Item 53 – PDF Output – Oct 2018

A bug was fixed where the height of the Cross Section shown in the PDF did not correspond to the drawing (graphic) that was being shown.

June 2019

Item 54 – Text/Font Size in Design/Panels Tab

When running Vespa2 on some laptops, the smaller screen and high resolution resulted in the Text Boxes on the Loading/Design/Panels screen to be enlarged. We have reduced the font sizes in these Boxes to appear more reasonable on a smaller screen. This is more of an issue with how Windows handles the fonts.

Left: 0.00 m
Right: 3.00 m
Base: 188.54 m
Top: 190.95 m
Height: 2.41 m
- Exposed: 2.17 m
- Embedded: 0.23 m
- Design: 2.12 m
Alpha: 56.82°

Item 55 – AASHTO 2015 Slope Weight

For the slope above the Wall, the Soil Type has been assumed to be Reinforced/Infill. More typically, we see that the actual Soil Type in this area is more similar to the Retained (Native). We have changed this due to user feedback.

Item 56 – Addition of Multi-Depth Gravity Walls

Vespa 2 now includes the ability to generate and analyze Multi-Depth Gravity Systems in general conformance to the NCMA Design Methodology.

Multi-Depth Systems can be selected from the list of preloaded Products in the Wall Unit Tab (for Licensors that have provided this information), or, they can be defined as "User Defined" Systems in the Settings/Wall Units menu.

To select a Preloaded Multi-Depth Set (System), go to the Wall Unit Tab, select the Licensor and Product Line, then click on the "Select Multi-Depth Set" toggle.

Only the Preloaded Systems that are Multi-Depth will now be shown in the Wall Unit Drop Down Menu. In the "Settings" Tab that appears to the right of the Drop Down Menu, a couple of options are provided as shown below.

In the settings, we can see that for this particular Multi-Depth system, there are two units being utilized (one is 925 mm deep and the other is 500mm deep). First, you set the "Wu Factor", which is an empirical limit you can apply to the maximum height any block can go as a gravity wall based on the block depth. For example, if the block depth is 2.0ft, and the Wu Factor is "2", the maximum allowable height that block could go is 4.0ft. Second, you indicate which of the Blocks should be used for the Geogrid sections in the Wall. The Multi-Depth Engine allows you to have both Gravity and Geogrid Sections within the same wall. As such, you can set which block should be used for the Geogrid Sections here.

Once the system is selected, if you plan on using Reinforcement for some sections, go to the Reinforcement Tab and select your reinforcement as before.
In the Soil Conditions Tab, the Drainage Layer is defaulted ON for a Face Drain. You must input a unit weight of Drainage Soil (Default is provided), as this is used in the calculation of your Wall Weight.

After you input your stations and go to the Panels Tab, you will see that a Preliminary Multi-Depth layout is provided (shaded portion is the larger Unit at the bottom of the Wall). This preliminary layout is only based on the Maximum Wu values you set in the "Settings" and is NOT A DESIGN.
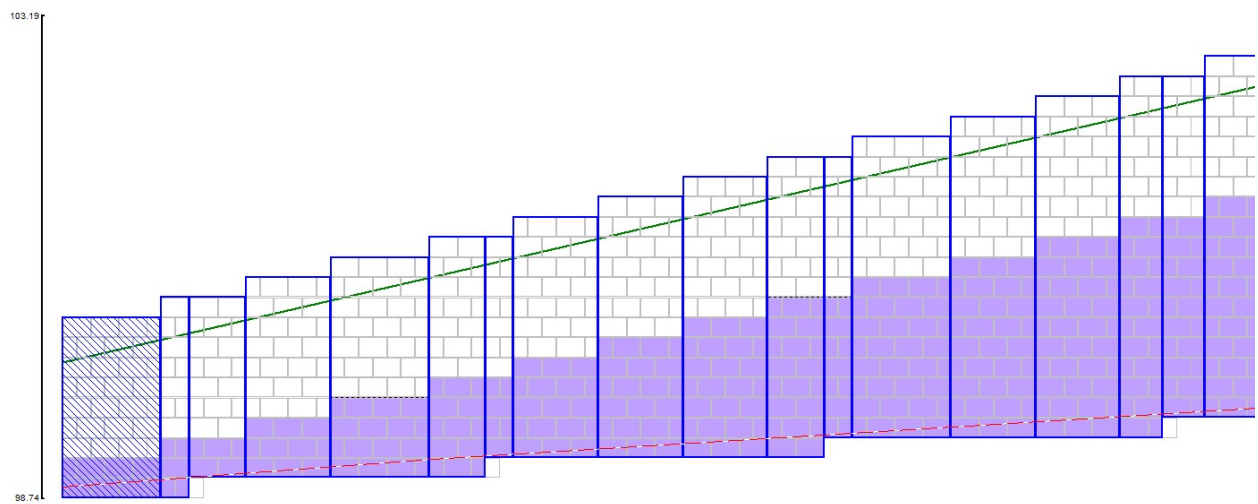
In the Loading Conditions, we see the Cross Sections, which contain a mix of smaller and larger units as shown below.
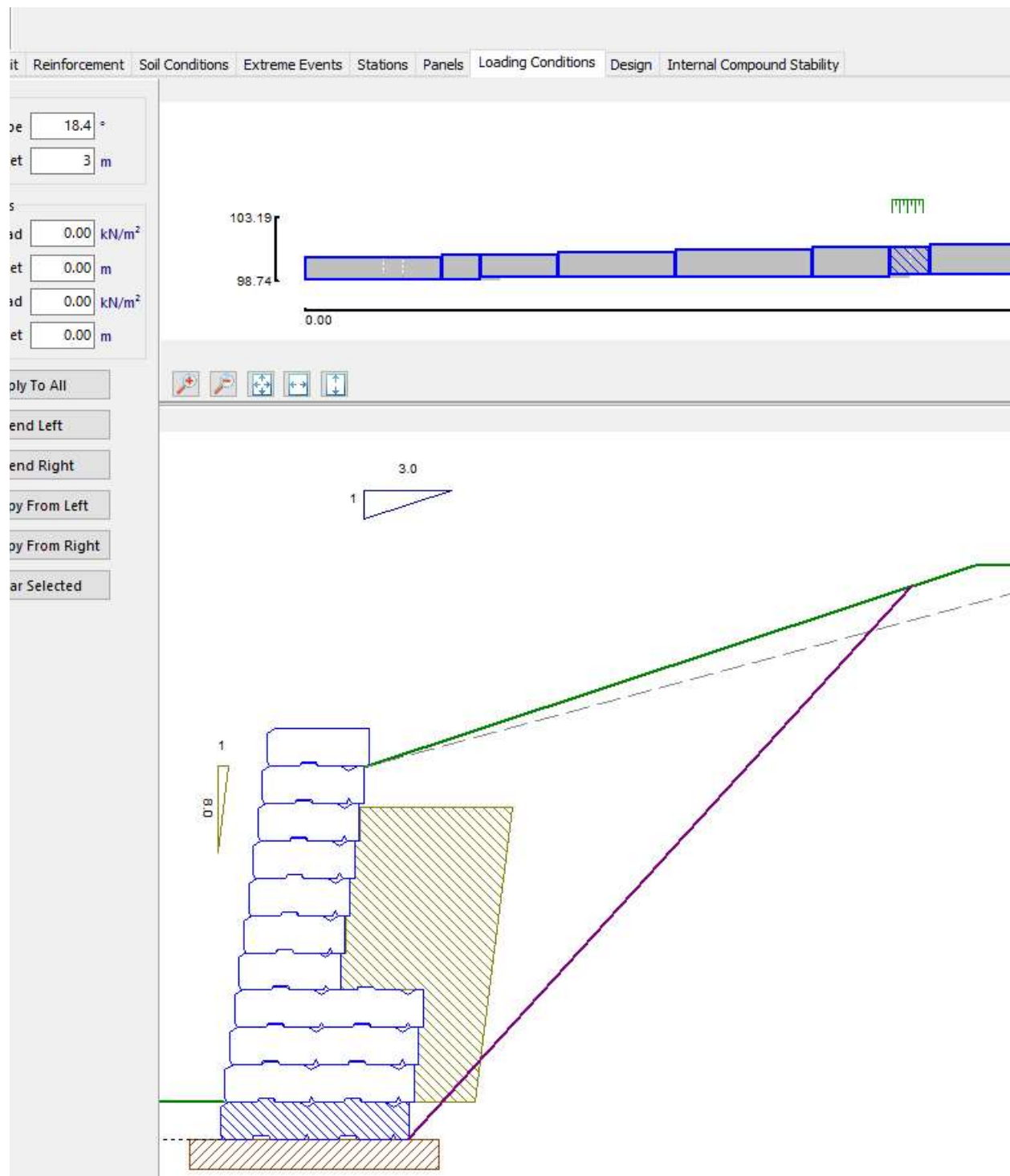
Again, this layout of smaller and larger units is not a Design, just a preliminary layout. Include your loading as before and move to the Design Tab.

In the Design Tab, the default for the "Generate" function is to generate a Multi-Depth gravity Wall. If you need this particular section to be a Geogrid Reinforced Wall, set the option to "Force Reinforced". If not, leave it unchecked. Note that if you select multiple panels in the Design Screen, and the Force

Reinforced option is ON, it will design those panels as Geogrid Sections, and leave the other Panels as Gravity Sections. However, if the "Generate All" button is selected, the Generate function will attempt to design the sections as Gravity Walls, but if they do not pass, it will switch to Geogrid Sections if a Reinforcement is selected.
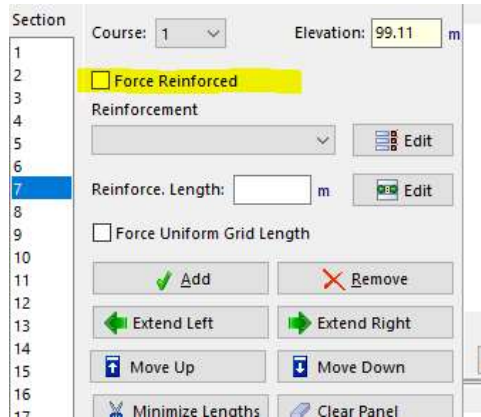
The Generate function for Multi-Depth Walls works as follows. First, it checks to see if the section will work using the largest unit, for the full height. If this does not pass, unlike a Grid reinforced Wall, nothing further can be done and it will just show the failed result. (A Grid reinforced wall can increase the length, number or strength of grids in an attempt to pass. A Multi-Depth Set is limited to the maximum depth of the largest block in the set).

If this first test passes, the Engine works for the top-down. Using both the empirical limit of Max. Wu and full analysis calculations, it determines the maximum height of the first depth of block at the top of the Wall. In this solution, it has determined that the smaller block can achieve 7 courses at the top of the Wall.
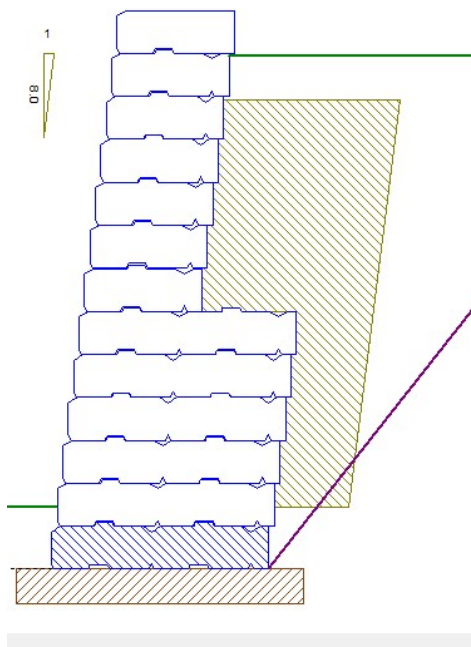
Next, it checks the Total height of the Wall again, using the Drainage Material unit weight as part of the upper portion of Wall. Note that this is different from the first check, where the entire Wall height was assumed to contain the Largest block. If the Total height does not work with the new arrangement (partial Drainage material as weight), then the engine continues to add larger blocks until it does.

If the system had three different Block depths, it would run one more iteration in the process.

In the Full Output, we can see that the calculations show a FS for each mode of failure, at each level (small block, large block). There is only 1 FS for Bearing Capacity, as it is for the entire structure.

Detailed Results			
⦿ Static ◯ Seismic			
Failure Mode	Layer	Course	Value
External			
✓ Bearing Capacity			4.80
✓ Overturning			3.10
✓ Overturning		7	3.33
✓ Base Sliding			1.69
✓ Base Sliding		7	34.77
✓ Minimum Embedment %			13.6600 %
✓ Minimum Embedment			265.0000 mm

In the above example, we chose a "Multi-Depth Set" that contained two different Block depths. The new MD Engine also has the ability to stack a single Block Increment to any depth

To create a User Defined For a Multi-Depth System, you have to first create the block as shown above (following steps 1-4 above for all depths of your Multi-Depth System. For example, if your MD System has 3 depths, you would create three separate blocks, such as "System A 300", "System A 600", and 'System A 900". These three blocks would represent three different depths (front to back depth) within the same system. You could add Geogrid Connection properties to these blocks as well if you wish.

Once the blocks are created, go to the new Multi-Depth Tab on the right side. In this Tab, you have two different options to create a Multi-Depth System.

Option 1 – Set Unit Sizes
This option is for a system that has between 2 and 3 Units that have a set depth (i.e they are manufactured to a set depth). We call this a Multi-Depth Set.

| Properties | Connection Properties | Profile Coordinates | Multi-depth |

A Multi-depth definition can consist of either a single selected Wall Unit allowed to repeat a number of times on each course, or as a set of two or three Wall Units of varying depth to be placed singlely on each course.

If multipleWall Units are chosen, the largest depth unit must be selected first, it will determine the allowable selections for the other units. The Wu Factor is a default only and can be changed by the user, and determines the maximum height allowed for that unit.

Multi-depth is only applicable to the NCMA Methodology.

Siena Stone 1425 - 39 inch
Siena Stone 1425 - 48 inch
Siena Stone 333 Multi-Depth
Siena Stone 925 - 39 inch
Siena Stone 925 - 48 inch
Sonoma Stone 750

New
Save
Cancel
Delete

Name of Set
Siena Stone 1425 - 39 inch

Base Unit

Maximum Units / Course
1

Intermediate or Top Unit   Wu Factor
                                                       2.50

Top Unit   Wu Factor
                                                       2.50

Select "New" and enter the Name of the Set. Then, Select your "Base Unit", which is the largest (deepest) unit, your Intermediate Unit (middle depth) and your Top Unit (or just the Intermediate/Top Unit if you only have two different depths in your system).

The Wu Factor on the Right side of the Intermediate/Top Drop Down Menu indicates the maximum height of Wall that unit can be used for, as a function of the block depth. For example, if I have a block depth of 500mm, and a Wu Factor of 2.5, the program will not stack that unit more than 2.5 x 500 = 1.25m in height. This is also how it determines the initial layout of the wall.

Then hit Save.

Option 2 – Incremental Multi-Depth Wall
If you want to create a MD system that uses a single block (module) to create various Wall depths (by placing them immediately behind eachother), use option 2.
In this option, we name the "set", just as above, then select our Base Unit from the Drop Down. In this case, it would be the single block that we are using the incrementally create the Multi-Depth Wall. Once

the Base Unit is selected, we set the Maximum Units/Course value to greater than 1. This value represents the maximum number of these units you would ever use to create a multi-depth structure. For example, if our Base Unit or Module is 500mm in depth, and I know that I would never place more than 4 of them back to back (for a depth of 2.0m), this number would be 4.

When this value is set to anything above 1, the Intermediate/Top Unit Drop Downs disappear, as the system now knows this is an "incremental" multi-depth system, not a "Set" System described above.

Item 57 – April 2020

New Graphic Interface.

A new DirectX Graphic interface was implemented, which solved the sizing issues users were having with small (laptop) screens. In the past, graphics generated on smaller screens where not handled properly, and became distorted or flipped. The new Graphics system allows Vespa2 to be run on almost any size of screen, while maintaining the integrity of the images. The DirectX interface is also much cleaner, less pixelated, and clearer than the previous version.

Item 58

AASHTO 2015 Revised to Include Live and Dead Strip Loads

For a full set of calculations, please contact support@ctiware.com.

Item 59

AASHTO 2020 Simplified Method and Inclusion of Live and Dead Strip Loads

Item 60

AASHTO 2015 and 2020 – Modifications to Calculations

a. For AASHTO 2020, the Coulomb earth pressure coefficient is now allowed, which utlizes the Wall batter (omega), even within the previous 10 degree minimum. Also, the External Interface Friction Angle is calcuated as the minimum of 2/3 x the Reinforced Internal Friction Angle or Retained Internal Friction Angle. Previously, this value was set to the External Beta. For a full set of Calculations, please contact support@ctiware.com.

b. For AASTHO 2015, in cases where a Broken Back Slope existed, we based the calculaton of equivalent Beta, or I, on the diagram shown below (AASHTO 11.6.3.3 and 11.10.5.5). As seen, even in the case where the Slope broke within the reinforced zone, a value for Beta External could still be greater than 0 degrees (based on 2H). For External Stability, a more accurate approach to modelling this would be to set the Beta External equal to zero, as there is no "actual" slope beyond the reinforced zone in this case. As a result, this will reduce the External Ka value, and associated Forces.
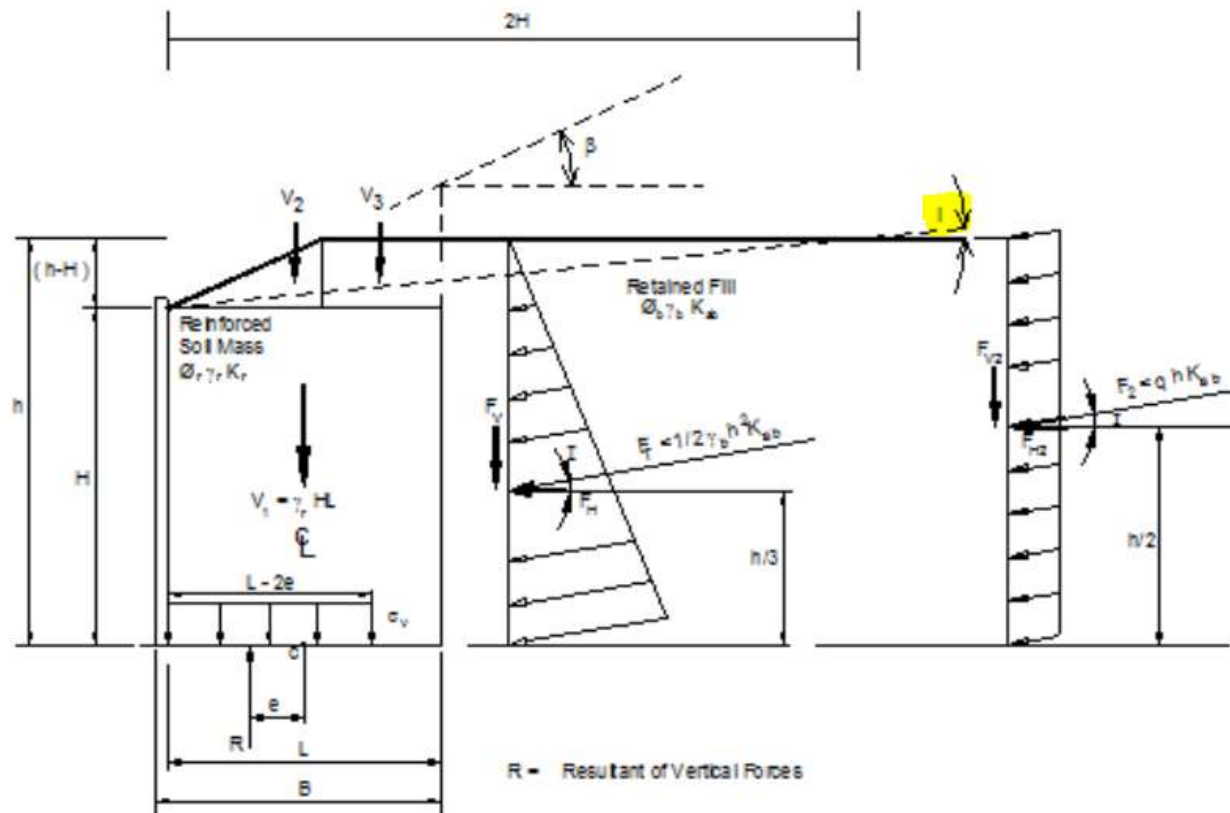
c. For Base Sliding Resistance, the conservative assumption was originally made to reduce the Foundation Cohesion by the Friction Factor, mu. This assumption was incorrect, as these modes of resistance are independent of eachother. The result was overly conservative Sliding Resistance values in the situation where Cohesion was used. This has been fixed.
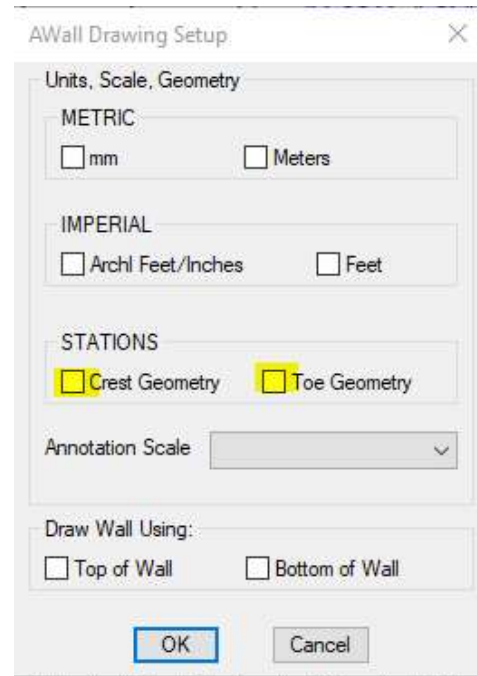
Item 61

Introduction of Awall2 and new Vespa2 Plan Import Function

Based on user feedback, the new Awall2 has been upgraded as follows.

1. Crest and Toe Geometry Inputs

At each Station, the user can now input Crest and Toe Elevations and Offsets, which will then be imported directly into Vespa2.
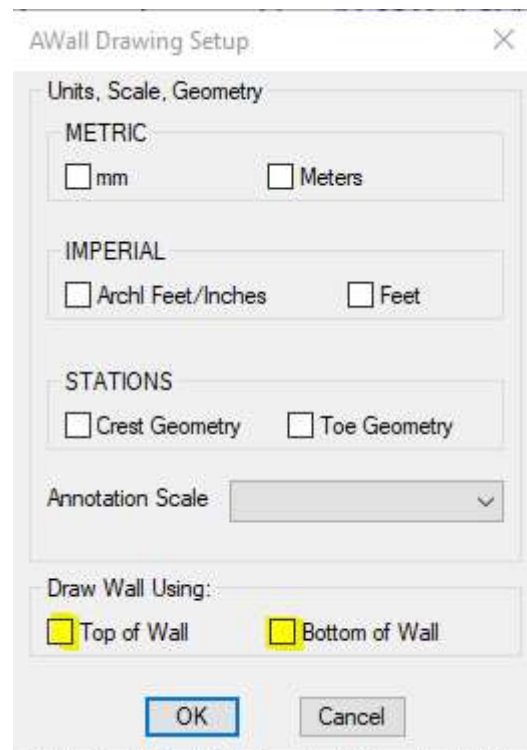
2. Option of Drawing the Top or Bottom of Wall Line.

By selecting either Top of Wall or Bottom of Wall, the user can specify which direction the Wall is projected when the "Create Wall" function is used.

The "Top of Wall" option is the way Awall has always operated. This assumes that the "Baseline", or line you draw to represent the Wall, is the Front Edge of the Cap Unit on the Wall. As such, the Create Wall Function projects "Down" from this to create the outline of the Wall Batter.

The new "Bottom of Wall" option now assumes that the line you draw to represent the Wall is the actual Bottom of Wall, where it intersects the Grade in front. As such, the Create Wall function projects "Up", to create the Cap Unit at the Top of the Wall.

3. Plan View Import / Export

When the Export Function is used in Awall2, the "Plan View" of the Baseline is now included in the File. As such, when the Station Data is imported into Vespa2, a new Tab is created which accurately shows

the true Plan View of the Wall. When the Wall is then Designed, whether using Gravity Blocks (Single or Multi-Depth), Geogrid Reinforcement, or No-Fines concrete, the Plan View is updated to represent the true footprint of the Wall, include block depth, wall batter, depth of reinforcements, and/or excavation limits. This Plan view can then be exported back to CAD (DXF) to overlay on an existing plan. This feature is very useful to determine potential conflicts with the Wall/Reinforcement and the Site (property lines, other structures, easements, etc).

Item 62 –

Plan View Import/Export updated.

Item 63 – EV/LL Option added. For all AASHTO Design Methodologies, an option was added to use the Vertical Earth Pressure Load Factor (EV) for Internal Tension calculations as opposed to individual Load Factors for each potential Load (Earth, Live Load, Dead Load).
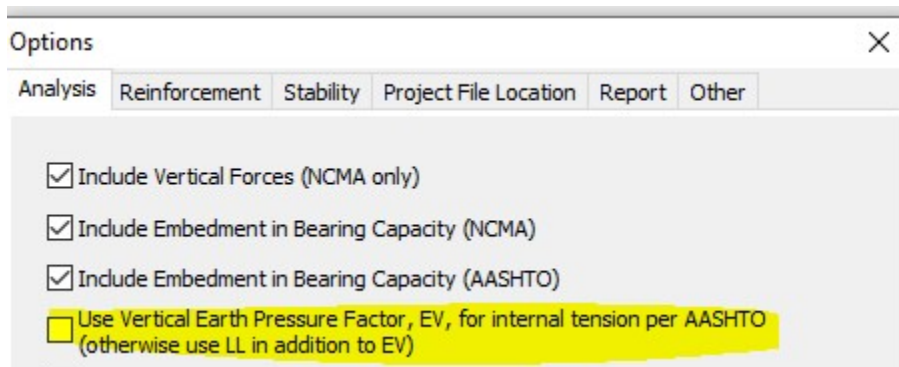
Item 64

AASHTO - Eccentricity Check presented as CDR value as opposed to binary check.

Item 65 – When large Toe Slopes present, Cross Section display was not centering image. Fixed.

Item 66 – DXF output – Applied Bearing Pressure labels were not being output when Wall mixed Multi-Depth Gravity Structures with Grid Reinforced Walls. Fixed.

Item 67 – NCMA Multi-Depth Gravity Analysis – New Method of inputting System.

A new method has been added to enable the user to add Multi-Depth Gravity Wall systems that are composed a combination of Facing Elements, and Tie-Back Modules, such as a Crib Type System.

In Item 56 (above) we discussed the addition of Multi-Depth Gravity Walls. To now include Crib type Wall systems or other "Tie-Back" Gravity Wall Systems, a new method of entry has been provided. **NOTE: The Gravity Analysis checks External Stability of the Wall ONLY. Internal Stability of the Wall is not considered and would be the responsibility of the System Manufacturer. This would include internal shear/tension between Tie-Back Members and Dead Man or Facing Elements, etc.**

Under Settings/Wall Units you will see the regular Dialogue Box for Facing System Maintenance. In the Properties Tab you create the **Face Element** first. This is done by selecting User Defined from the Licensors Drop Down Menu. Select "Block" as the Unit Type then enter the Properties (dimensions, unit weight, shear capacity, etc), then hit "Save".

Once the Facing Element has been defined, you can now create a Tie-Back or "Extender" Unit. This type of unit can only be used as a Multi-Depth component, and cannot be used in a Wall by itself. To do this select New again, then "Extender" from the Block Type.

Now input the Extender (Tie-Back) properties. The % Hollow Core accounts for the infill volume in the Crib, so this must be determined in advance. Note, the Face Height must be the same as the Facing Element previously defined to be used together. Note that the Shear Capacity of the Extender must be specific to this unit, not that of the Facing Element.

Then hit Save.

You can add an additional Extender Unit with a different Depth as well. For example, if you had a second type of Tie Back that was 600mm in Depth, this would be added in the same way.

Go now to the Multi-Depth Sets Tab. Select "New" and name your new Crib System, then select "Multi-Depth with Extenders". Now you can select the Face Unit you just defined. Below this you have the option of selecting up to 2 different Extender Units. You must select the Shorter one first, then the Longer one. In our case, we just have 1 Tie-Back or Extender unit, so we will select it in the Shorter Unit Drop Down Menu. Then hit Save.

If we create a new Project and select the User Defined, Multi-Depth Set we just created, the Wall will first Panelize using the Facing Element only as shown here.
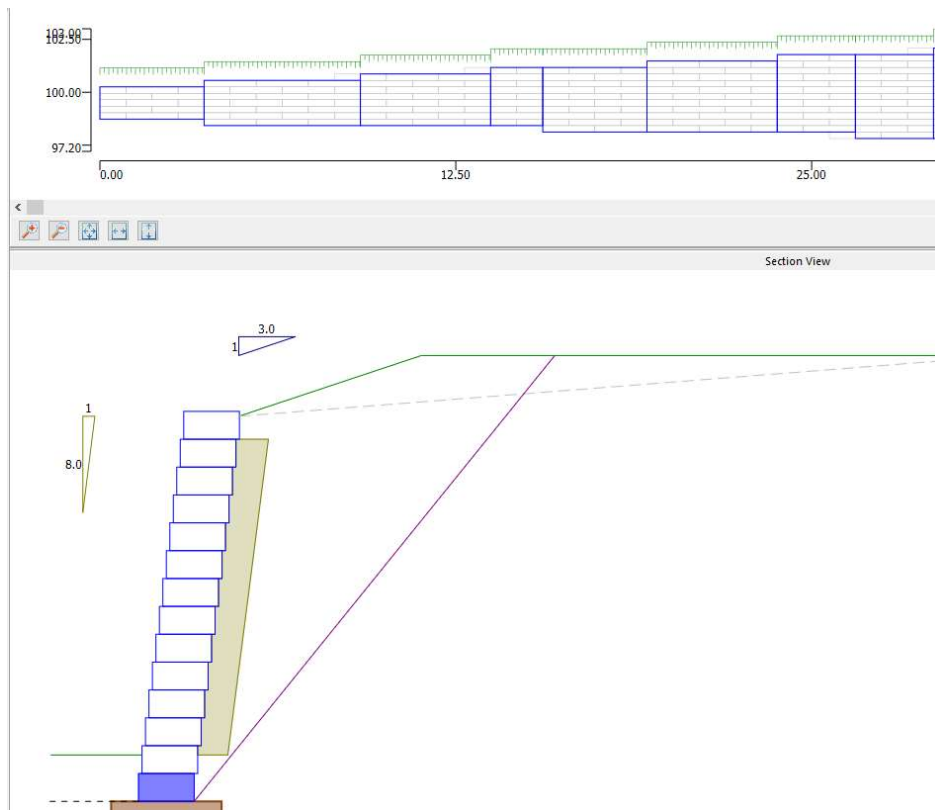
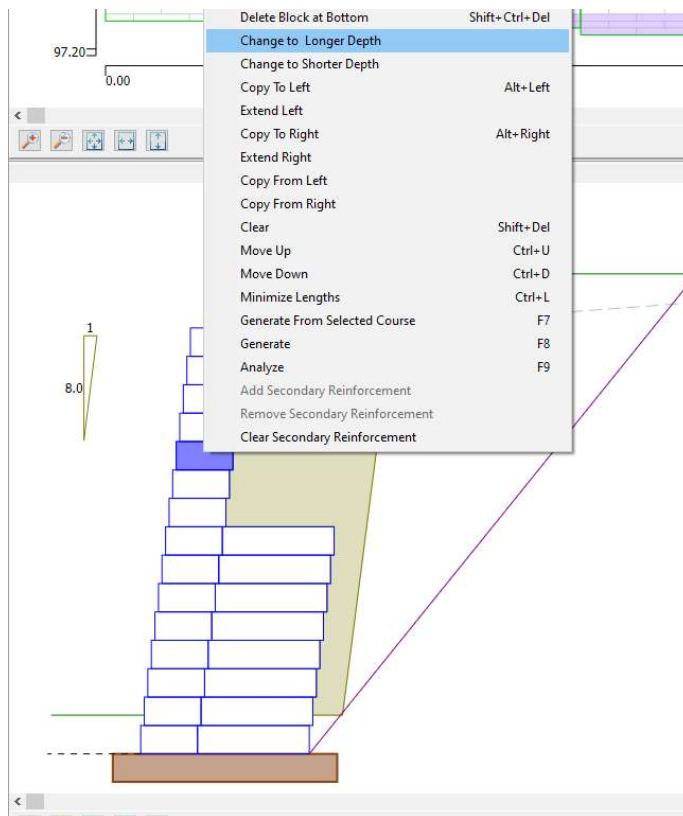In the Design Tab, if you select Generate All, Vespa will analyze each Section and add Extenders as necessary to meet the minimum FS required.
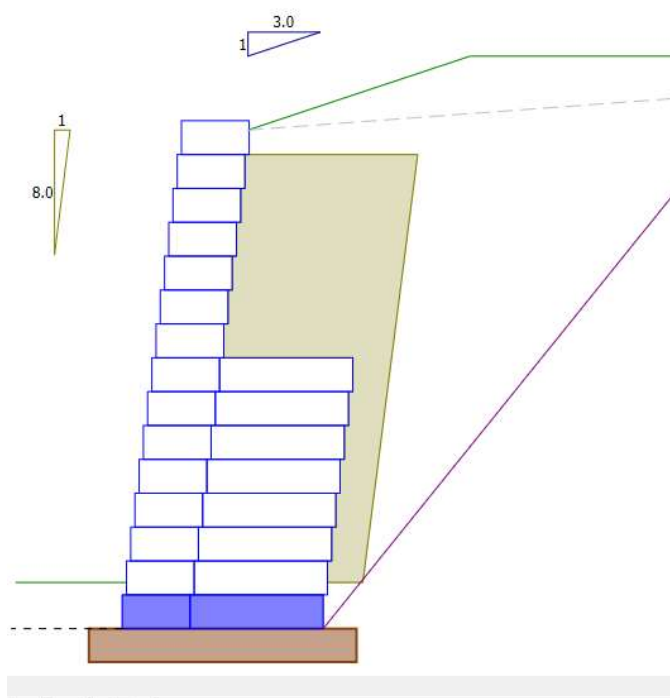
You can manually add Extenders at any Course by clicking on that Course, then Right Clicking to bring up the Menu. In the Menu you will see an on option to "Change to Longer Depth" or "Change to Shorter Depth".

This will result in adding a Longer Depth to that course and all those courses below it. Note that it starts with the Shortest Extender.
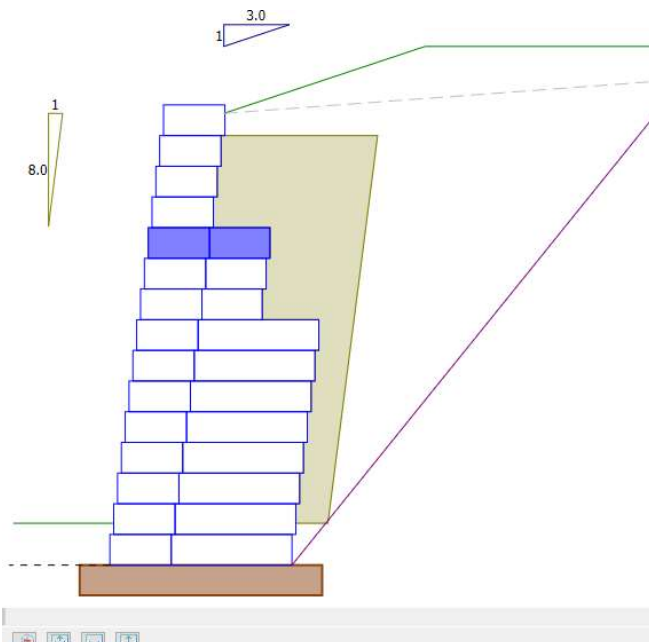
This process can continue by simply selecting the Longer or Shorter Depth again from the Right Click Menu.

In the Quantity output, the number of Facing Units, Extender 1 and Extender 2 are provided.

Item 68 – AASHTO 2020 Internal Resistance Factors

From AASHTO 2015 to AASHTO 2020, the default values for the Resistance Factors for the Internal Stability Analysis changed as follows.

Pullout –   0.9 (2015) / 0.7 (2020)

Connection –   0.9 (2015) / 0.8 (2020)

Tensile Overstress –   0.9 (2015) / 0.8 (2020)

The default values did not update automatically in the Design Criteria Tables and the user was required to modify them. This has been corrected and the default values now update to reflect the changes.

Item 69 – AASHTO External Stability: Horizontal and Vertical Components

As noted in Item 60 above, *for AASHTO 2020, the Coulomb earth pressure coefficient is now allowed, which utlizes the Wall batter (omega), even within the previous 10 degree minimum. Also, the External Interface Friction Angle is calcuated as the minimum of 2/3 x the Reinforced Internal Friction Angle or Retained Internal Friction Angle. Previously, this value was set to the External Beta.* While we did incorporate this change into the Ka calculation, we conservatively kept the External Beta to determine the Horizontal and Vertical Components of the Lateral Soil/Surcharge Loads. AASHTO allows use of the Interface Friction Angle in place of External Beta, however, the current representation of the AASHTO equations do not account for the Wall batter (omega). To properly account for the Wall Batter, we have adopted the following modified version of the Horizontal and Vertical force calculation.

AASHTO 2020 –

$$F_H := 0.5 \cdot K_{ab} \cdot \gamma_b \cdot H^2 \cdot \cos\left(\delta_e\right)$$

$$F_V := 0.5 \cdot K_{ab} \cdot \gamma_b \cdot H^2 \cdot \sin\left(\delta_e\right)$$

AASHTO 2020 (Vespa) –

$$F_H := 0.5 \cdot K_{ab} \cdot \gamma_b \cdot H^2 \cdot \cos\left(\delta_e - \omega\right)$$

$$F_V := 0.5 \cdot K_{ab} \cdot \gamma_b \cdot H^2 \cdot \sin\left(\delta_e - \omega\right)$$

We believe this is the intent of the Equation, however, it has not been explicitly shown, likely due to the application of many vertical MSE Walls (Panel/Steel Reinforcement) represented in this document.

Item 69 – Secondary Reinforcement

This version includes the option to include Secondary Reinforcement (SR) in the Wall. At this time, the SR is only used for quantity estimation purposes and visual representation. The SR is not included in the Wall Analysis. Some users include smaller layers of geogrid near the top of the Wall for local stability against handrail and fence loads, for example, which is the main purpose of including the SR in this form. We do provide the option of placing small layers of SR between layers of PR, as this does have some beneficial effect on reducing Connection Loads, however, it is not accounted for in the analysis as noted.

To access the Secondary Reinforcement Option, go to Settings/Options/Reinforcement Tab. You will see a toggle for "Enable use of Secondary Reinforcement".
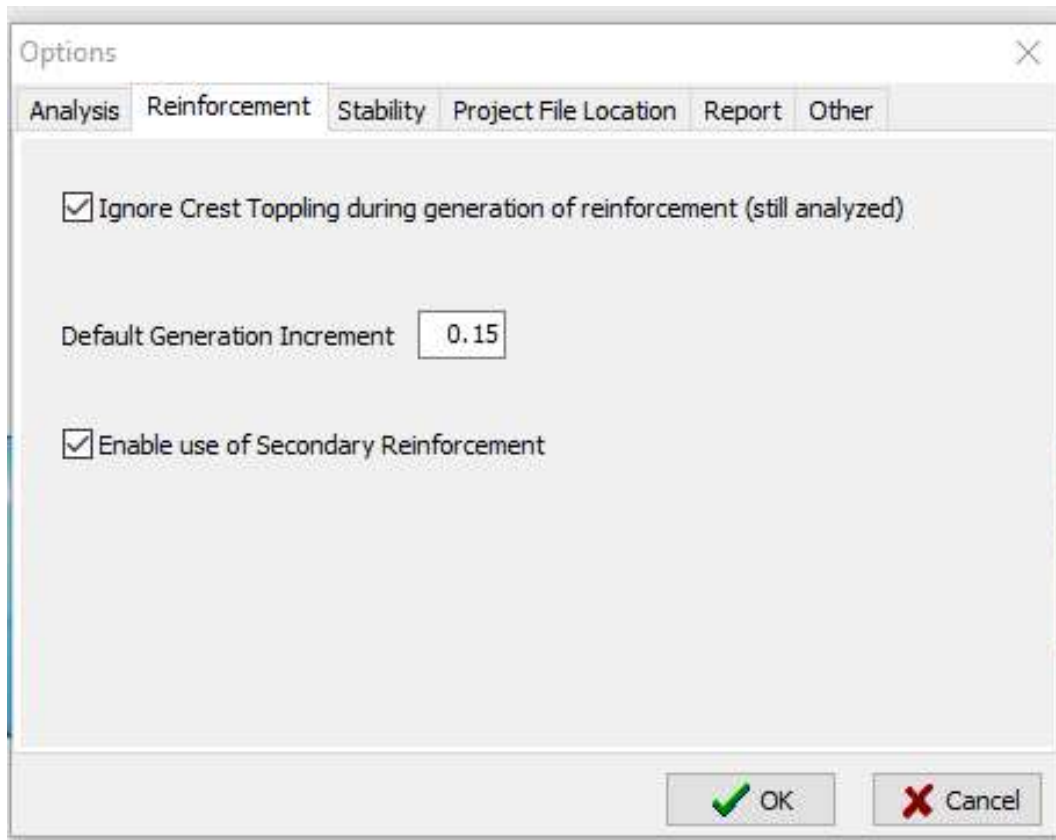
When you enter the Reinforcement Tab in your project, you will now see the option of setting a particular geogrid as the "Secondary Reinforcement". This geogrid can be used as the primary as well.

In the Design Tab, you can run your normal "Generate" routine to establish the Primary Grid. Remember, the analysis does not consider the Secondary Grid, so the Wall has to work using only the Primary Grid first. Once the Wall is designed, you have 3 options for applying the Secondary Grid; Manual, One/Primary Interval, and Every Available Course.

If you are just putting a small layer of Grid at the Top of Wall for example, you can select Manual, then enter the Grid Length.

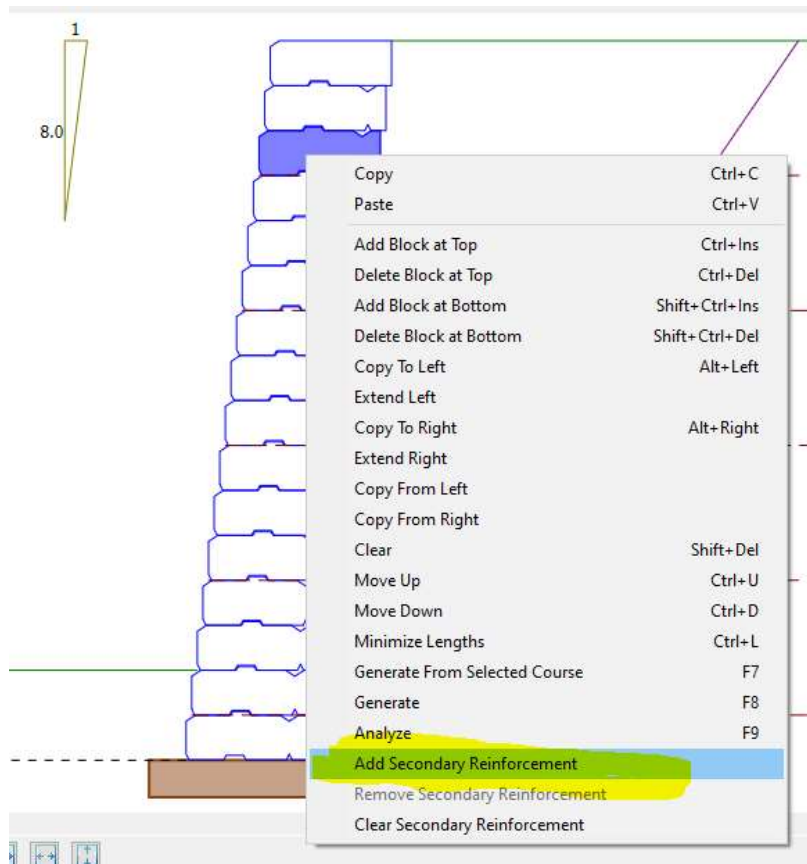You would then click on the Block that you want to put the Grid on, then Right Click to bring up the Menu. Then Select "Add Secondary Reinforcement"

Copy	Ctrl+C
Paste	Ctrl+V
Add Block at Top	Ctrl+Ins
Delete Block at Top	Ctrl+Del
Add Block at Bottom	Shift+Ctrl+Ins
Delete Block at Bottom	Shift+Ctrl+Del
Copy To Left	Alt+Left
Extend Left	
Copy To Right	Alt+Right
Extend Right	
Copy From Left	
Copy From Right	
Clear	Shift+Del
Move Up	Ctrl+U
Move Down	Ctrl+D
Minimize Lengths	Ctrl+L
Generate From Selected Course	F7
Generate	F8
Analyze	F9
Add Secondary Reinforcement	
Remove Secondary Reinforcement	
Clear Secondary Reinforcement	

The reinforcement is then added. You can Copy this placement to all Panels by selecting "Copy to All".
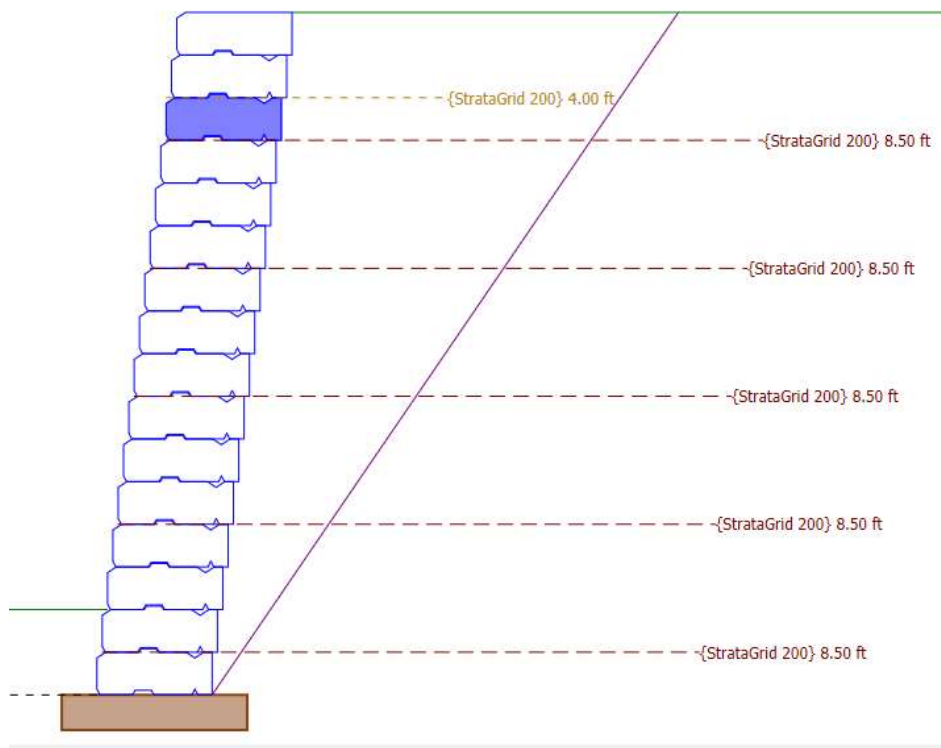
If you would like to place a SR layer between all primary layers, select One/Primary Interval, and one will be placed between existing Primary layers.
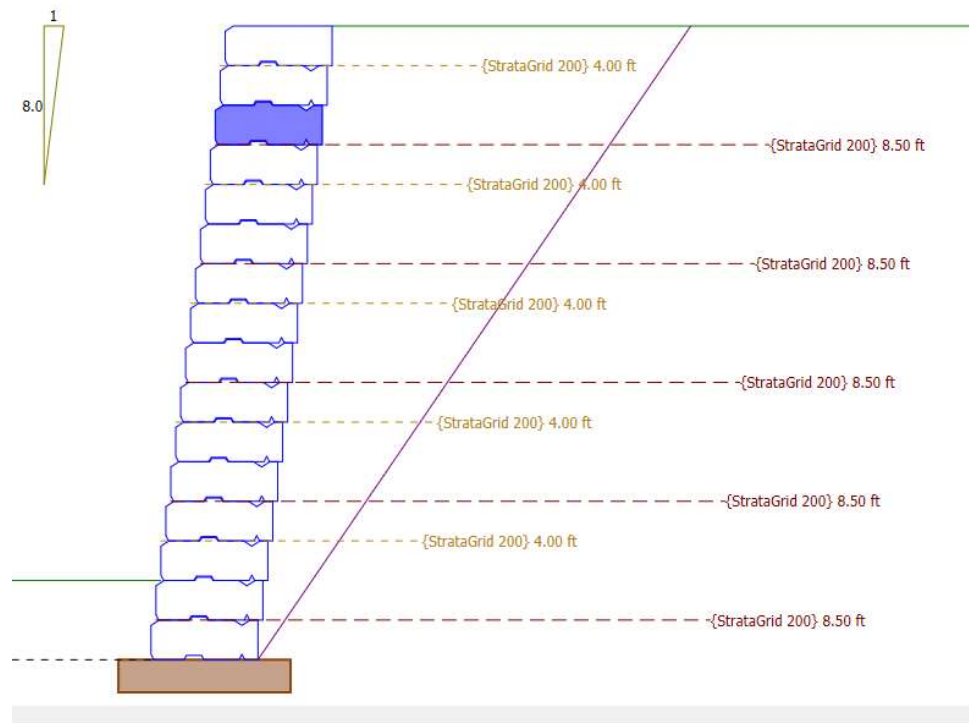
If the final option is chosen, a SR layer will be placed on every available course.

We plan on expanding the use of SR as more is incorporated into future Methodologies.

Item 70 –Internal Compound Analysis – **Important Update**

For the ICS Analysis, it was determined that a bug existed in the code that resulted in the over-estimation of the contribution of Grids Forces for Static Analysis. *Through a parametric analysis, on average the F.S. was overstated by 2%-10% for high walls (>15ft) with steep slopes and/or a combination of a slope with surcharge loading (Broken Back), <u>although many factors will affect these numbers</u>.* **This has been fixed.** We have now provided additional methods of modelling ICS more accurately. In the Settings under Options/ICS, we have the option of setting the grid orientation relative to the tangential angle of the failure surface. When the ROR value is on, the grid is tangential to the failure plane. AASHTO recommends using the ROR for extensible (geogrid) reinforcements when RC=1, unlike Steel Strips or Matts for example. As an additional measure, we have provided for the inclusion of cohesion in the Retained Zone. As always, caution must be used when applying cohesion in any slope stability analysis.

It should be noted that the ICS calculation is not a substitute for a proper Global Stability Calculation conducted by a qualified Geotechnical Engineer. The Geotechnical Engineer is responsible for the final, complete Global Stability Analysis "around and through the SRWs" (Refer to Secton 3 NCMA Manual, 3rd Edition).

Item 72 – Multi-Depth Analysis

Discussion of Assumptions and Addition of Swale Option

The Multi-Depth Gravity model is based on the NCMA Conventional Analysis. As the NCMA does not explicitly deal with Multi-Depth Gravity Structures, the basic NCMA model was followed, with the following modifications.

The standard Conventional Model assumes a fixed unit depth and material unit weight. The Multi-Depth model is a combination of Concrete Units (filled or solid) and Drainage Backfill as shown below.


The initial Model allowed for 3 Unit Depths (Sets), like what is shown above. Typical Unit Depths might be 1.0m, 0.5m, and 0.3m, for example.
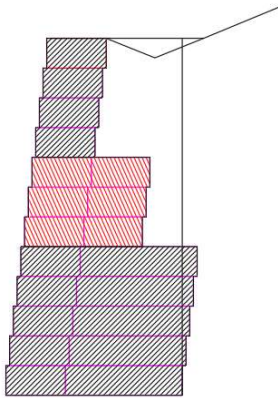
The Multi-Depth model followed the NCMA conventional approach where the Wall height at the back of the Wall was equal to the H. This assumption was based on the fact that even with relatively high Gravity Walls, given the wall batter, limited depth relative to height, and standard Swale geometry, the Hext is only nominally different than H. However, with the introduction of the of the Crib/Extender option, the potential for deeper and higher walls exists. Therefore, to model the Multi-Depth Gravity geometry more accurately at the top of the Wall, an option has been provided to include a Swale.

When running a Multi-Depth analysis, in the Design Tab you will see an input field called "Swale Width". The NCMA Best Practice Guide recommends a minimum of 0.9m (36") for the Swale Width (Refer Chapter 4 – Water Management and General Design Considerations). If this is selected and a value is input (1.0m for example), the analysis is modified to account for this new Geometry.

The Swale will be shown as a dashed line to indicate it is considered in the Analysis as shown below.
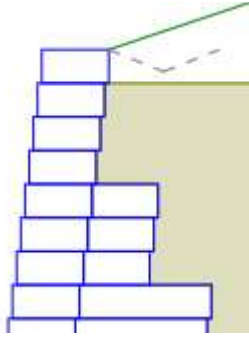
For the Extender Option, it must be noted that the analysis is limited to External Stability (Base Sliding, Overturning, and Bearing), and Internal Shear Resistance.   As many of these Crib/Extender systems are proprietary in nature, the Internal Stability of the Connections between units (such as Tie-Backs for example), Tensile and Moment Resisting Capacity of members within the system, are the responsibility of the Vendor and are not checked or verified in the analysis.

An additional Interface Shear check has also been added to the Top of the Base Course. Previously, we checked each new Tier for Interface Shear, then the Base Course was the traditional Base Sliding Check. The assumption was the bottom of the Base Course (sliding along the gravel base) would always have less sliding resistance than the top of the Base Course, as this would typically have some mechanical shear connection (key/groove) as well as friction. As this may not necessarily be true in all cases, the additional check was added.

Item 73 – Multi-Depth Analysis Copy/Paste Functions

When designing a complete Wall, we have added the ability to Copy/Paste and Extend Left/Right the Multi-Depth configuration. The configuration of MD Units is taken from the "Top-Down", as shown below. As with the Geogrid Extend Left/Right and Copy/Paste function, the new sections still must be Analyzed independently as they have not been verified yet.

Item 74 – Dec 2021

In the October 2021 release, an additional field was added to the Soil Conditions Tab called "Earth Surcharge". It was a minor addition and as such an explanation of it was not included in this document. For clarity, it allows the user to input a different Soil Unit weight for the Slope above the Wall than

assuming it is the Infill Material. This weight is used for Weight of Wall calculations. If it is left blank, Vespa will default to using the Reinforced Unit Weight.

Item 75 – AASHTO 2020 and 2015 (Dec 2021)

In April 2020 (Item 60 above), after discussions with various State DOTs, it was determined that the empirical limit of 10 degrees for the Wall Batter (<10 degrees considered vertical) was at the discretion of the Designer. Applying this limit resulted in a model that was inconsistent as the Wall Batter was used to accurate define the geometry of the structure (Weight of Wall, Moment Arms for Resisting Moments, etc), but was ignored for the calculation of Kab, Internal Failure Angle. Vespa allowed the application of Omega consistently throughout the analysis. However, we have just added an additional option of setting Omega to zero if the user decides to. Refer to Settings/Options/Analysis/AASHTO.

Item 76 – ICS Updates

76.1 - In Settings/Options/ICS the user can set the Factor of Safety for Static and Seismic. The default is set at 1.3 for Static and 1.1 for Seismic (NCMA Defaults). The field was not set up to allow the user to input a decimal (i.e. 1.5), and only allowed round numbers (i.e 2). This has been rectified to allow 1 decimal place.

76.2 – When selecting multiple panels (Shift Key+Select) and running ICS, the results for all panels were not being saved as you moved from Tab to Tab (i.e. you would need to re-run the analysis to see the results for each individual panel).

76.3 – An option has been added in Settings/Options/ICS to allow the user to either include Block/Grid Connection in accordance with Figure 8-6 (NCMA) or include only Block/Block Shear (with or without reinforcement at interface).

76.4 – Full Output button has been added to ICS Screen.

January 2022

Item 77 – AASHTO Methodology. ICS option had been turned off when running AASHTO. It has been turned back on.

Feb 2022

Item 78 – The Earth Surcharge (Soil Tab) input field that was added (to change the Unit weight of the Slope above the Wall) required the back-end database to be updated to load older files. This was done.

Item 79 – NCMA Equation 7-29 which calculates the reduced Geogrid Length used for Bearing Resistance calculations was updated to account for Wall Batter per the latest printing (5th) of the NCMA Manual.

Item 80 – Updated Help File added.

March 2022

Item 81 – Multi-Depth Auto-Generate upgraded for more optimum solutions for Full Wall.

November 2022

Item 82 – NCMA, Seismic. The Option to use the External Horizontal Acceleration Coefficient for Seismic Crest Toppling was reversed. Therefore, if it was "on" the Internal would be used (more conservative) and "off" the External would be used. This has been fixed, so "on" uses the External Coefficient and "off" uses Internal.
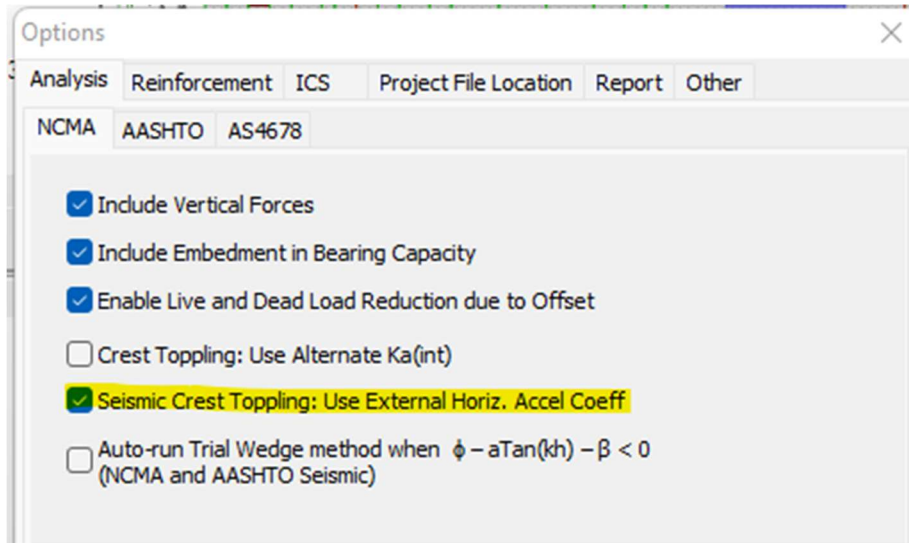
Item 83 – No Fines Concrete (Gravity Wall) Update (NCMA / BS8006) Feb 2023

When utilizing the No-Fines Concrete, users have indicated that in some cases the Gravel Base is extended fully under the No-Fines Backfill as shown below.
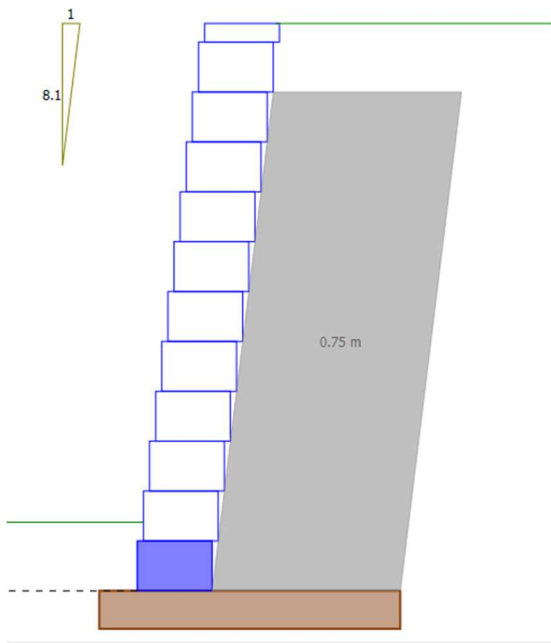
In other cases, the Gravel Base is only under the Block (Facing) and the No-Fines Concrete Backfill is bearing on the foundation (native) material.

To accommodate this, a check box has been added to the Wall Unit Tab where the user can indicate which approach they are utilizing. When checked ON, the Align to No Fines option extends the Gravel Base under the entire No Fines backfill zone (as shown above). In this scenario, the Base Sliding resistance is calculated in the same way as a typical conventional wall where the Base Material Friction Angle is utilized along with the Masonry Friction Factor (μ). It should be noted that for a smooth, manufactured masonry block, the μ value recommended by NCMA is 0.7 (Well graded gravel base). As the No-Fines Concrete is placed/poured directly on the gravel base, it is likely that the coefficient used could be higher. When the option is OFF, it is treated more like a Reinforced Mass and the Foundation Soil friction angle is used, along with the option for Cohesion.

Item 84 – AASHTO 2020 Conventional (minor update).

To account for Live and Dead load offsets, a factor is calculated that reduces the applied load as a function of the offset distance (Qlfactor and Qdfactor). This conservative factor is based on a 1H:1V line of influence. The equation compared the Load Offset to the wall Height (H). To better accommodate the effect of a broken back slope, the equation now compares the Load offset to the wall Height (H) + the slope height (hs).

Item 85 – British Standard

The British Standard now includes the No-Fines Analysis. Also, Creep and Durability reduction factors to the Connection.

Item 86 – AU Standard

Minor modifications to Fg(n) calculation.

Item 87 – Standard Soil Parameters

For the Unit Weight of Gap-Graded Gravels (GP), Vespa currently uses 125 pcf (19.6 kN/m^3). The range typically found in available tables/resources varies between 110-130 pcf, however, some users have noted that it is typically closer to 115 pcf. As such, we will adjust the default to 115.

Also, the friction angle provided for GW Gravel is currently 39 degrees. While this is often the case, some users note that Geotechnical Reports may not allow this high of a Friction Angle without testing. As such, we have adjusted it down to 37 degrees.

Note that these values (Unit Weight and Friction Angle), are just a starting point based on some typical known values. The user must modify these values based on available information and have them verified by the Site Geotechnical Engineer.

Item 88 – Alternate Seismic Method – NCMA (Update to Existing Feature)

Section 9.2 of the NCMA Design Manual, 3rd Edition, states that the calculation of the horizontal seismic coefficient is restricted to conditions where kh < (tan (ϕ-β). When conditions are such that this restriction is exceeded, Vespa provides two options. In the Settings/Options/Analysis/NCMA, there is an option to turn ON/OFF the Alternate Seismic Method. If this is OFF, Vespa runs the Seismic Analysis by simply setting the term ϕ- atan(kh) -β = 0, with the warning shown below. This approach appears to be consistent with the NCMA Software, although not technically correct.

For external Seismic Analysis (NHI Eq. 7-6), the term ϕ – aTan(kh) – β has been set to 0. This modified solution is not technically correct and should be carefully evaluated by an engineer.

If the Alternate Seismic Method option is ON in the Settings, then for External Stability a Trial Wedge Method of Analysis is run. This method must have a Crest Offset defined. In this update, we have added the ability for the user to specify the Trial Wedge Angle at the back of the Reinforced Zone to start the analysis process.

For Internal Stability, the Alternate Seismic Method applies the approach outlined in NHI-10-024 which calculates the weight of the Active Wedge, then calculates the dynamic inertial force, Pi. The update to this version allows for displacement effects to reduce the kh(int) in the same way that it reduces kh(ext), which is Ao x 0.5.