LECWall 3 - Concrete Insulated Wall Panel and Column Design and Analysis

Table of Contents

Introduction

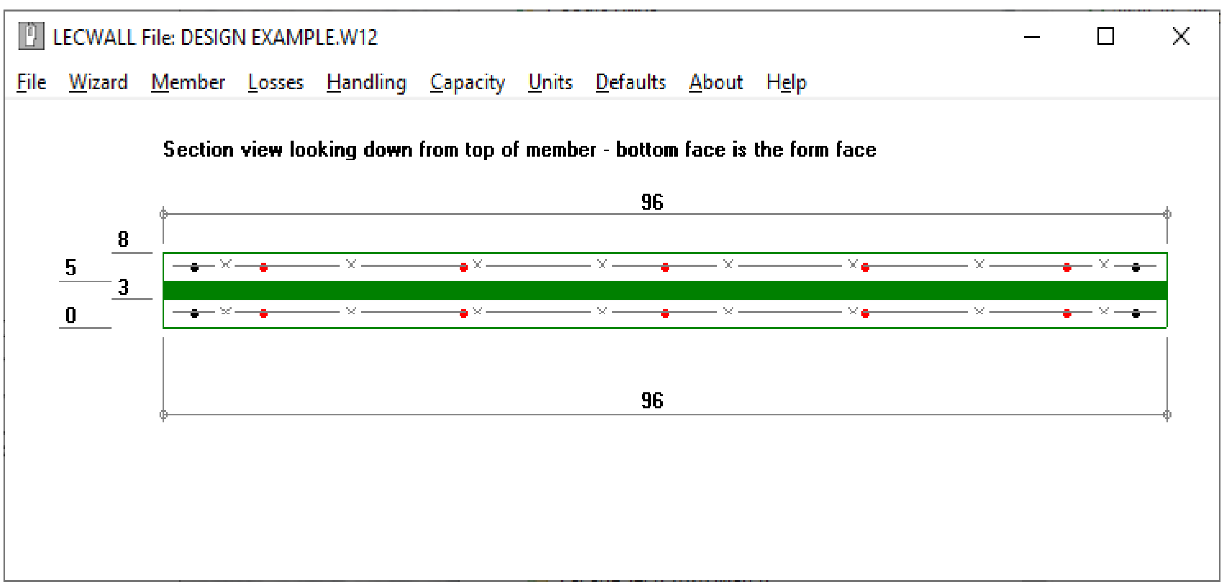
<Back>

LECWall is the industry standard for the design and analysis of concrete insulated "sandwich" wall panels. It was written by engineers for engineers. It can analyze prestressed and/or mild reinforced wall panels with zero to 100 percent composite action. Flat, hollow-core and stemmed configurations are supported. Complete handling analysis is also included. It can analyze multi-story columns as well. This program is meant for use by a licensed engineer proficient in the design of precast and/or tilt-up concrete wall panels and columns. Some features are:

- Complete handling analysis under service and ultimate conditions with two or four point pick and user specified form and impact factors.
- Partial composite action between wythes can be either user specified from 0% to 100% or calculated directly using a choice of built-in partial-composite analyzers.
- Up to 16 openings or notches/reveals of any size accommodated at any location.
- P-Delta slenderness and temperature effects combined with wind, earth pressure and gravity load analysis under service conditions or at ultimate strength using ACI 318-14/19 or NBCC 2015.
- Calculates bow due to eccentric prestress force.
- Autocheck of capacity vs. applied at 100 points along member. Critical locations are flagged.
- Automatic or manual prestress loss calculations
- Will handle wall panels or tied columns with any combination of mild reinforcing and strand.
- Up to 10 lateral supports with optional spring constants.
- Hung panel option
- Easy to use graphical interface with diagrams and on-line help screens.
- Choice of English or Metric units.

The Methodology section of this help file describes the codes and algorithms used, as well as any assumptions made by the program. This section is useful as documentation and support for your wall panel or column design calculations.

Wall problems are given a unique name by the engineer and saved on the hard drive in any directory. These problem files can be specific cases or masters for commonly used section types. Use the Save File As... command in the File pull-down menu to save a master as a specific case.



The main window will display a dimensioned section view of the currently loaded problem, complete with reinforcing and insulation. The window size can be adjusted by "grabbing" the edge of the form and moving it while holding down the left mouse button.

The steps involved in creating a new section and running the analysis are:

1. Select New File from the File pull-down menu. If another wall file is loaded you will first be asked if you wish to save it.
1. Select Project Info from the **Member** pull-down menu. Fill in the appropriate identifying information for this run (optional). Notes will appear on the first page of the Input printout.
2. Select Section Dimensions from the **Member** menu. Fill in the section dimensions by using the mouse or Tab key to select an input box and type the desired value in the box. Click **Done** to close the form or **Cancel** to start over.
3. Select Materials from the **Member** menu. A form will appear listing concrete and steel properties. Some boxes will already contain default values as a convenience.
4. Select Reinforcing from the **Member** menu. There are three categories of reinforcing to fill in: strand, rebar and/or welded wire fabric.
5. Select Openings from the **Member** menu if there are any openings or notches in the member.
6. Select Connectors from the **Member** menu if wythe connector properties need to be input for partial composite calculation.
7. Select Reveals from the **Member** menu if there are any reveals or depressions on the front or back face of the member.
8. Select Additional Weight from the **Member** menu if there are any significant areas where insulation is cut out or any concrete haunches are present.
9. To examine or change calculated prestress losses, select Prestress Losses from the **Losses** menu.
10. The input file can be saved at any time by selecting Save File from the File menu.
11. Select Handling Check from the **Handling** menu to check stresses and ultimate capacity at stripping, yarding/bunking, during transportation or at erection. Select **Print** to print input and handling graphs (three pages of input plus up to four pages of graphs).
12. Select Capacity Check from the **Capacity** menu to check the member's in-place ultimate capacity and stresses under design loads. Input applied loads, initial bow and support locations. Click **Max Tension Stress Envelope** or **Max Compression Stress Envelope** to find the critical load cases and the critical points along the span. Select each critical load case (1 thru 14) then click on **Show Stresses** to check for a cracked section. Click on **Print Diagrams** to print the moment and stress charts, if desired.
13. Select Interaction Curves to check capacity at critical points for all fourteen load cases. The **Autocheck** button can be used to find critical locations automatically. Move the slider bar to display the values at each critical point. Click on **Print Diagrams** to print a one-page report showing the location of the critical section and interaction curves for pressure and suction at that point.

Note **-** you can select the "**Wizard**" from the pull-down menu to cycle through these steps in order. (Click on **Next** or **Back** at the bottom of each form.)

Methodology

<Back>

Section Properties:

Section properties of multi-wythe composite members are calculated using the modulus of elasticity of the **Main Structural Wythe** (as designated by the user in the Section Dimensions window). If the modulus of elasticity of the secondary wythe differs from the main wythe, the secondary wythe area and moment of inertia is multiplied by the modular ratio between the two wythes. See the Partial Composite section for a description of how partially composite properties are determined. Section properties are calculated at 100th points along the member.

Openings & Reveals:

Section properties are automatically modified at opening and reveal locations. Openings are cut all the way through one or both wythes, while Reveals are depressions in either the front or back face of the member. **To ensure accurate section properties, openings should not overlap with other openings or reveals, and reveals should not overlap with other reveals or openings.** The program will not check for overlaps. That needs to be done manually by examining the plan views provided in the *Opening* and *Reveal* design windows (be sure to update the views first by clicking the **Update** button).

Any strands, bars or WWF that cross through openings are cut and development lengths are adjusted automatically. Click on the Section button in the Capacity Check window to verify the strand, bar or WWF areas at 100th points. The bar or strand areas are reduced accordingly if the reinforcing is not fully developed at the section in question.

Strand Development Length:

The standard ACI method is used. The calculated development length can also be altered by using a multiplier, if desired. To do this, click on **Setup**, select Defaults and change the values at the bottom of the screen accordingly. A value of 1.0 at ends and 2 (or higher) at openings is recommended, to account for the differing strains near the opening.

Prestress Losses:

The PCI Design Handbook is used for calculation of initial and final Prestress Losses in the member (Section 5.8).

Handling Analysis:

Vertical lift lines with two cranes or a spreader beam are assumed to be used for stripping members from the bed. **Note: If inclined lines are used for lifting, you need to manually calculate the additional moment generated by the angle of the inclined lines and account for it in the member design.** See Handling Check for more information.

The center of gravity of the panel is listed near the top of the *Handling* window. Use the C.G. to assist in specifying the lift point locations to provide a more equal distribution of lift forces, if desired. Potential panel tilt is also displayed. Tilt can occur when rollers are used for lifting to equalize the left and right reactions.

An ultimate capacity check for handling forces is also provided. Statically calculated unfactored moments are first multiplied by the user-specified handling (or impact) factor, then multiplied again by an ultimate load factor. This ultimate load factor is initially set to 1.4, but can be changed by clicking on the *Setup* menu item and selecting Defaults. Strain compatibility is used to determine the member's ultimate capacity in the horizontal position.

1.0 times the theoretical cracking moment can also be displayed, per ACI 318, Section 11.8.1.1c. By keeping the member capacity greater than 1.0MCr, additional capacity will be present after cracking, which should provide a "warning" deflection before failure occurs. According to the PCI Insulated Wall Panel Design Standard, if member flexural strength is at least twice the applied factored load, then the ultimate capacity can be less than 1.0MCr. Note that this criteria only applies at the location along the length of the panel where the onset of cracking is expected, not along the entire length.

P-Delta Analysis:

The program performs an iterative second-order analysis of the member under each of the fourteen specified load cases. The PCI 8th Edition Handbook procedure (Section 5.10.3.1) is used to calculate bow and P-Delta forces. The member is considered to be pinned at both ends for the analysis (a conservative assumption when rigid floor ties are used).

If stresses exceed the modulus of rupture (user specified as the "Cracking stress coefficient") at any point along the member, the section is assumed to be cracked and the analysis is repeated using cracked section properties. Cracked I is taken as 0.35Ig, per ACI 318 Table 6.6.3.1.1a. Alternately, an I-effective is used for the Beam-Spring analysis, per ACI 318-19 24.2.3.9. There is also an option to use a pre-cracked section, as is common with slender tilt-up panels. I-cracked is calculated using ACI 318 Eqn 11.8.3.1c.

Click on the Show Stresses button to check for cracking anywhere along the length for a specific load case. As a general rule, cracking should be avoided under service load cases for prestressed members.

Forces and deflections due to eccentric gravity loads, earth pressure and wind are calculated using a built-In frame analysis module. The force in the optional floor tie is the total of effects from the eccentric gravity load moment, earth pressure, wind forces and restraint of predicted thermal bow. The floor connection can be set to be a first floor, at grade connection. Floor tie forces can be somewhat unpredictable (they have been known to pull out occasionally). Therefore, a ductile design for the floor ties is recommended to avoid a brittle failure if overstressed. Use the Spring Constant option to model the deformation of the tie connections.

Calculation of Bow:

Member bow is calculated based on differential temperature strains, wind load deflection and member bow due to applied gravity and earth pressure loads. Bow due to non-concentric prestress force is included.

Compression members are usually stressed so as to minimize bow. Such bow is hard to predict accurately in real life situations due to other factors such as humidity, storage methods, etc. The **Bow & Temperature** section of the Capacity form allows manual input of the predicted initial bow, based on the designer's experience.

I-Effective and I-Cracked:

For single-wythe panels and dual wythe panels using manually input percent composite, I-cracked is used when fr exceeds 7.5 Sqrt F'c. I-cracked is set as 0.35 * I-gross, per ACI 318 Table 6.6.3.1.1a (except for Beam-Spring). The **Beam-Spring** method calculates I-cracked for each wythe separately, using ACI 318 Eqn 11.8.3.1c. I-effective for mild-reinforced panels is calculated using ACI 318-19 Table 24.2.3.5. I-effective for prestressed panels is calculated using ACI 318-19 24.2.3.9. The Capacity screen has an option for analyzing a **pre-cracked** section. This is useful for the P-Delta analysis of mild-reinforced, slender, load-bearing tilt-up panels, or precast panels that have a risk of cracking prior to erection. When checked, I-cracked is substituted for the middle 2/3 of the panel, starting at 1/6 times the length from each end.

Incremental cracked section analysis: The Beam-Spring method uses an incremental approach to determine I-effective for each wythe at 100 points along the member length (unless the *pre-cracked* option is checked, in which case a fully-cracked moment of inertia is used). Multiple frame analyses are performed, with the loading increased from onset of cracking until full load is applied. I-effective for each point is adjusted after each run to account for the stress (and stiffness) redistribution that occurs as the load increases and the cracked area spreads along the member. This provides a more realistic model of the stiffness variations present along the panel length for each load case. For example, when a tie-back connection is present, cracking at the tie-back could create a plastic hinge, altering moments and stresses.

Wythe connector stiffness is reduced in any cracked areas along the panel length (Beam-Spring only). This reduction factor defaults to 0.75, for a 25% reduction in stiffness. Wythe manufacturers may use a different factor in their plug-in modules, based on their own testing.

Ultimate Capacity Interaction Curves:

The member ultimate capacity is calculated at a point along the member length selected by the engineer. The strain compatibility method is used to plot the points along the interaction curves. The equations used are from the PCI Design Handbook, Fig. 5.10.1. A straight line is plotted from the transition point of PhiPn = 0.10f'cAg, where Phi = 0.65, to PhiPn =0, where Phi = .9. Alternately, if the *Use Canadian material resistance factors* option is checked (see Defaults), the program will use factors specified by the Canadian code, CSA A23.3. For composite members (with two wythes), only one wythe is used for the compression block.

1.0 times the theoretical cracking moment can also be displayed, per ACI 318, Section 11.8.1.1c. It shows up as a small red circle on the Interaction Curve diagram. By keeping the member capacity greater than 1.0MCr, additional capacity will be present after cracking, which should provide a "warning" deflection before failure occurs. According to the PCI Insulated Wall Panel Design Specification (PCI 150), this applies to non-prestressed walls only. Also, if member flexural strength is at least twice the applied factored load, then the ultimate capacity can be less than 1.0MCr. Note that this requirement only applies at the location along the length of the panel where the onset of cracking is expected, not along the entire length.

For insulated wall panels, if percent composite is input manually, partial composite capacity is estimated by first computing the capacity at 0% composite and again at 100% composite, then finding a ratio based on the percent composite specified (see Partial Composite). The 0% and 100% capacity curves are plotted along with the partially composite curve for comparison purposes.

If the Beam-Spring method is selected, percent of ultimate capacity is found by summing the connector resistances from the closest member end to the point selected. Maximum connector slip/strain is assumed at the end connectors. The percentage of tension reinforcement that can be transferred between wythes by the wythe connectors is then found by using a bi-linear connector strain distribution at each point along the panel. This then becomes the % composite for Ultimate value.

New File

<Back>

Use this command to clear the current wall file in preparation for the input of a new wall file from scratch.

Open File

Use this command to retrieve an existing wall file from disk. You will first be asked if you wish to save the currently loaded file.

Save File

Use this command to save the currently loaded wall file to disk.

Save File As...

Use this command to copy a wall file and give it a different name.

Print Input

This command will send a three-page listing of the input data with a section view and a plan view for the current file to the printer. Additional input data is printed with the Handling Check and Ultimate Capacity Check printouts.

Print All

Combines LECWall printed output in a form suitable for a PDF file. One file is created, instead of separate PDFs for Input, Handling, Capacity and so on. More information and an example can be found in the online documentation in the file titled "LECWall 'Print-All' Explainer".

Exit

Will ask if you wish to save the currently loaded wall file and exit to Windows.

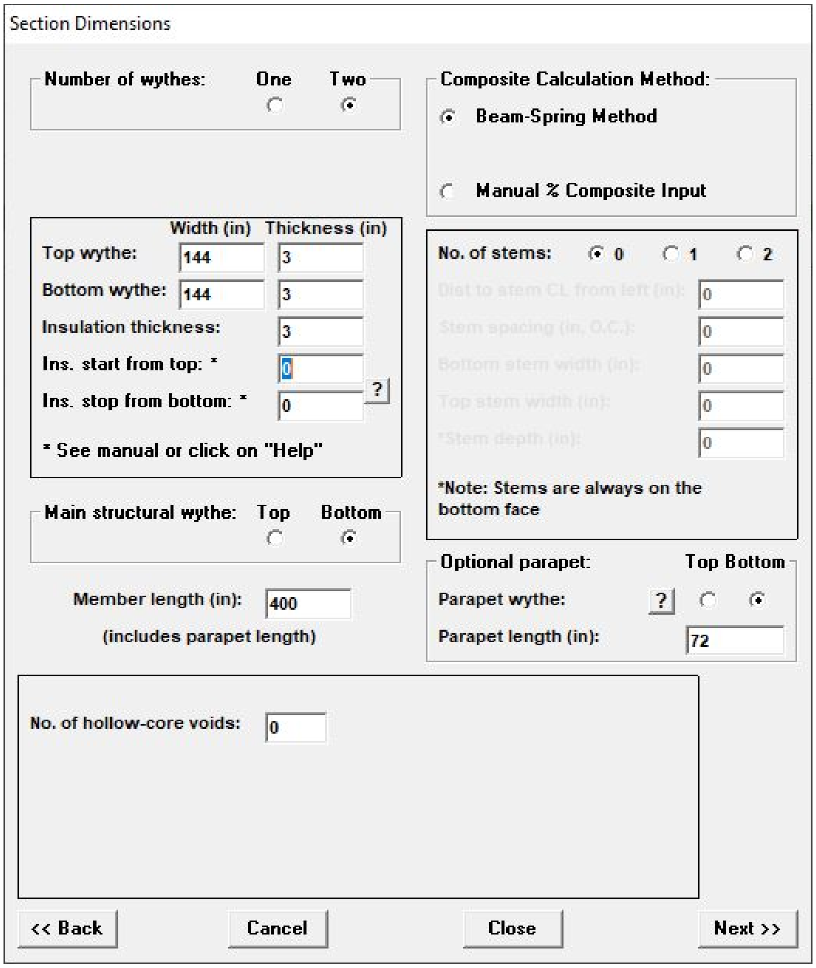
Section Dimensions

<Back>

This form contains the pertinent dimensions of the wall panel or column to be analyzed. If there is no top wythe, or you are modelling a column, set the number of wythes to *one*. If one wythe, the member can be designated as either a wall or column. Cracked section stiffness is assumed to be lower for walls vs. columns, per ACI 6.6.3.1. This gives more deflection for a wall. Walls use 0.25Ig and columns use 0.35Ig for the analysis.

The bottom wythe is required and cannot be zeroed out. Select the stiffer wythe as the Main Structural Wythe. The Main Structural Wythe defaults to the top wythe (screed face), since the bottom wythe (form face) often has reveals cut into the thickness. Stems and hollow-core openings can only be placed on the bottom wythe. In this case, the bottom wythe should be selected as the Main Structural Wythe, unless the top wythe is still stiffer. The program will warn you if the selected wythe is not as stiff as the other wythe. This is important when modelling percent composite or non-composite members, since stress values are higher on the stiffer wythe.

If the member is an insulated panel with internal ribs, it can be modelled as a single wythe member with rectangular hollow-core voids.



Insulation Start and Stop Points:

If a composite member has insulation which does not extend all the way to the top or bottom of the member, specify the insulation start and stop points in the boxes provided. The start point is the distance from the top of member (left end), or bottom of the optional parapet (also left end), to the point where the insulation starts. The stop point is the distance from the bottom of the member (right end) to the point where the insulation ends.

At those areas where insulation has been omitted (near top and/or bottom of member), the extra weight of the concrete replacing the insulation is accounted for and section properties revert to 100% composite, regardless of the percentage of composite action specified by the user.

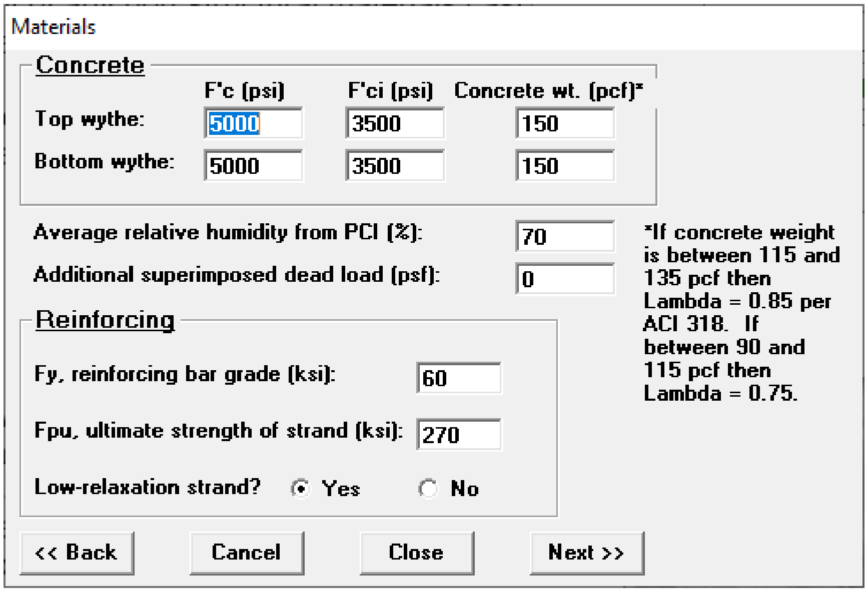
Optional Parapet:

The Parapet feature allows one panel wythe to extend above the other one. It can be the case where the panel inner wythe stops short and is used to bear roof joists. The outer wythe extends upwards above the roof line to form a parapet. This option is only available when using the beam-spring analysis method. The parapet length is included in the member length. The parapet wythe to be extended can be selected, either the top-in-form wythe or the bottom-in-form wythe. Note that the LECWall software does not check wythe connector tension forces. These can be critical at the base of the parapet. An example can be found in the online documentation in the file titled "LECWall Parapet Explainer".

The section drawing on the main form is updated when you click **Close**, **Back** or **Next**. Check this drawing to verify that the dimensions were input correctly.

Materials <Back>

This form lists the concrete and steel properties. Note that the top wythe can have a different unit weight and strength than the bottom wythe. If the Section Dimensions form has the number of wythes set to one, then the F'c and weight input boxes for the top wythe will not allow any input. For this reason, the Section Dimensions form should be filled out before the Materials form. (F'c = Concrete ultimate compressive strength at 28 days, F'ci = Concrete ultimate compressive strength at release)



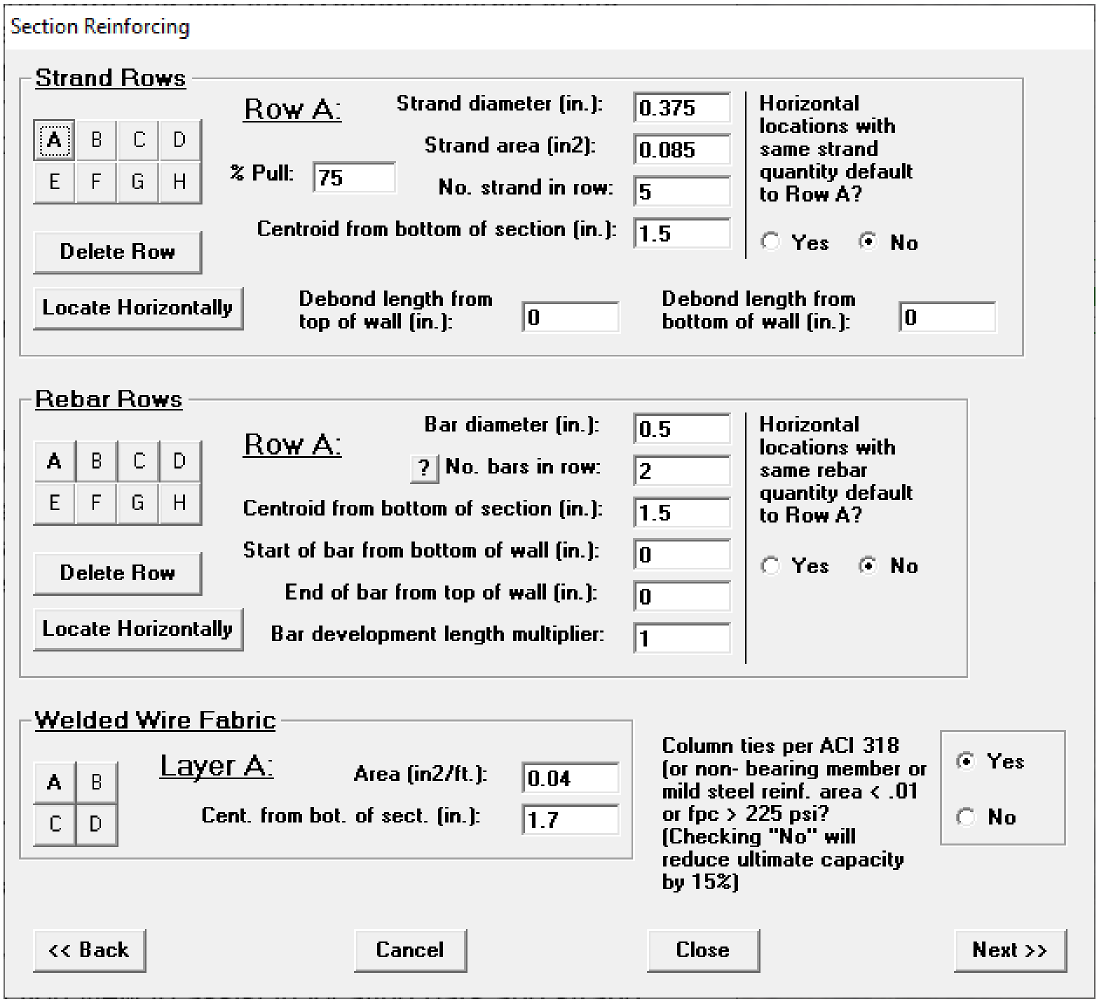
The *Additional Superimposed Load* input is useful for adding the weight of any non-structural materials cast in the panel, such as insulation, wythe connectors, solid areas, brick veneer, etc.

Low-relaxation strand is assumed as a default. Click on "No" at the Low-relaxation strand prompt if stress-relieved strand is to be used. This information is used to compute prestress losses.

Click **Close**, **Back** or **Next** to close the form or **Cancel** to abandon the **Materials** form input and start over.

Reinforcing <Back>

This form contains all the information related to the strand, rebar and welded wire fabric (WWF) in the section. Up to eight rows of strand or rebar and four rows of WWF can be specified. Click on button **A** to input Row A, etc. If more than eight rows are needed, combine rows and use the average centroid of the combined row. The bars or strand will be spaced uniformly across the section automatically. If a special spacing is used or there are openings that cross the reinforcing, click on **Locate Horizontally** to specify the actual bar spacing across the section. Locations are measured from the left edge of the panel in section view. To automatically adjust multiple strand and bar rows to a common spacing click "Yes" to the prompt "Horizontal locations default to Row A?"

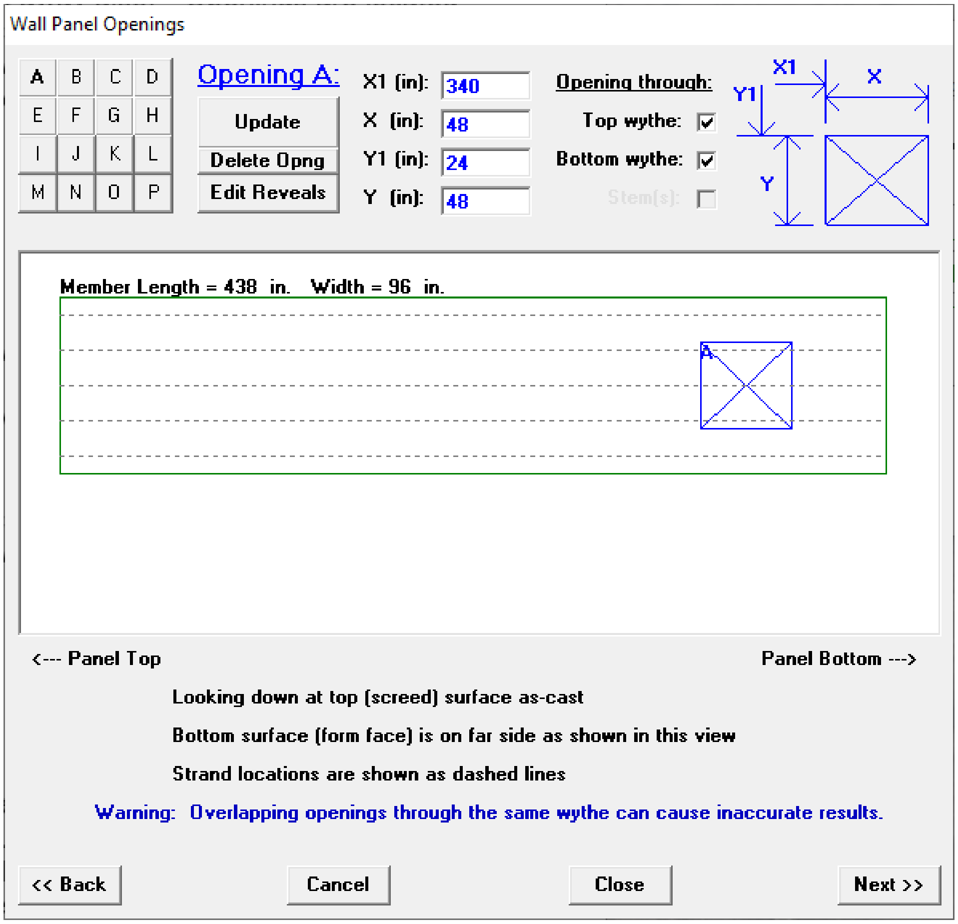


A value of 75 for **% Pull** is typical for low-relaxation strand (or 70% for stress-relieved). Note that reinforcing bars do not need to extend the full length of the member. Input a start point from bottom and from top, as desired. If a parapet is present (see Section Dimensions), the bar start distance from top should include the parapet length, for both the parapet wythe and non-parapet wythe. Strand debond lengths are input similarly - always include the parapet length for debond from top. See Sections to verify that the reinforcing placement is as desired.

Openings

<Back>

This form displays all openings and notches in plan view in blue. (Reveals are displayed in red for reference.) Up to sixteen openings can be specified (A-P). **Openings should not overlap with other openings or reveals.** Click to place a check mark in the boxes for each wythe the opening will penetrate and check the **Stem** box if the stem is cut away also.



Openings are located relative to the upper left corner of the plan view. Note that the section view is cut looking to the right. The top edge in the plan view is the left edge of the section view. Use the Sections view (in the Capacity screen) to assist in locating bars and strand to miss openings (also see Reinforcing - *Locate Horizontally*). Use the **Update** button to view any changes made to openings.

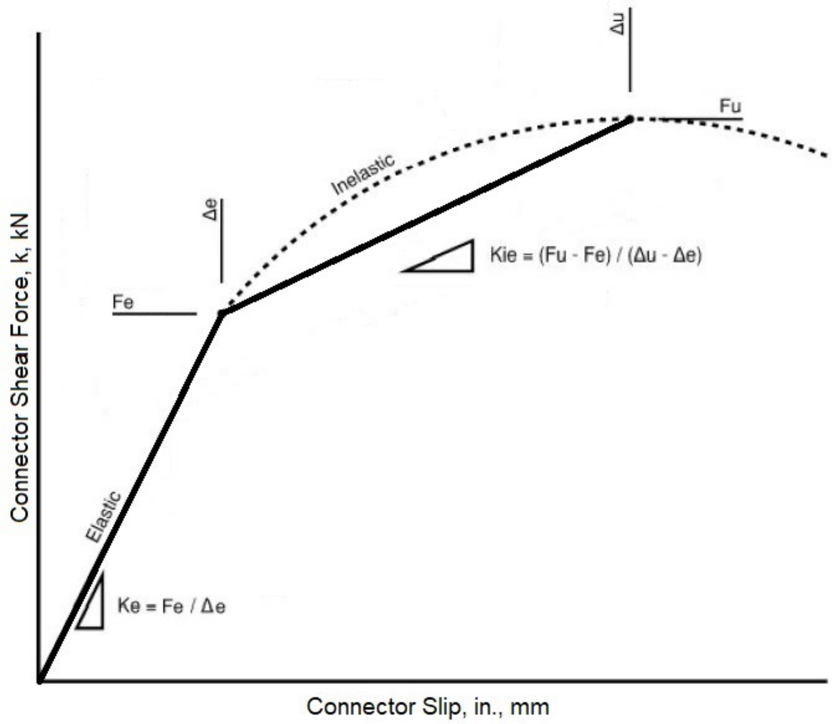
Wythe Connector Selection

<Back>

Specify the type of wythe connector and their spacing on this screen. There is a choice of manufacturer plug-ins or connector properties can be input manually. Four property values are needed for the manual input option (Fe, Fu, Δ-e, Δ-u). These values are per connector and correspond to key points on the generic strain graph shown. Most connectors behave elastically, in a linear fashion, until they reach an elastic limit. This is the point where the connector stiffness begins to decrease and slip increases. Connector elastic and inelastic stiffnesses (Ke, Kie) are calculated by the program from the four values provided. If a plug-in is selected, the boxes for the four force and slip input values are pre-filled and won't accept Input.

A factor of safety multiplier (phi-factor) is applied based on the failure mode and statistical variability. (This could vary from around 0.5 to 0.75.) Δ-e = Fe/Ke, Δ-u = (Fu-Fe)/Kie + Δ-e.

Some system manufacturers have provided their connector catalogs to us with the appropriate strain values. These have been incorporated into LECWall. This saves time and eliminates input errors. To use, select a Manufacturer Plug-In, then select a Connector Type. Only the connectors appropriate for the specified Insulation thickness will be displayed. Some manufacturers allow stiffer "secondary" connectors at the panel ends. Rows 1 through 3 can use these connectors. Additional connectors can also be provided in the end three rows. This allows for a higher connector concentration where the shear is highest.



$$K_E = \frac{F_E}{\Delta_E}$$

$$K_{IE} = \frac{F_u - F_E}{\Delta_U - \Delta_E}$$

Where: K_E = elastic stiffness

K_{IE} = inelastic stiffness of plastic stiffness

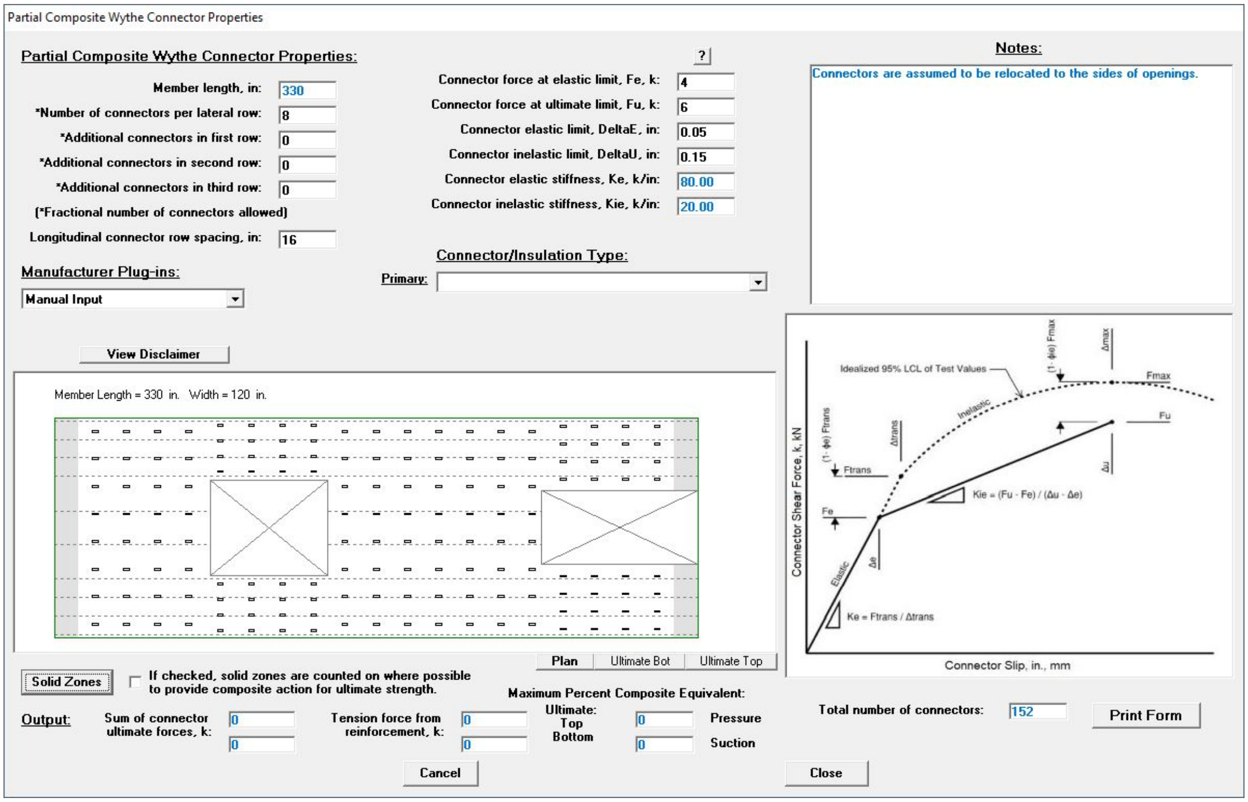
F_E = elastic load limit

F_u = ultimate capacity or peak load

Δ_E = deflection corresponding to the elastic load limit

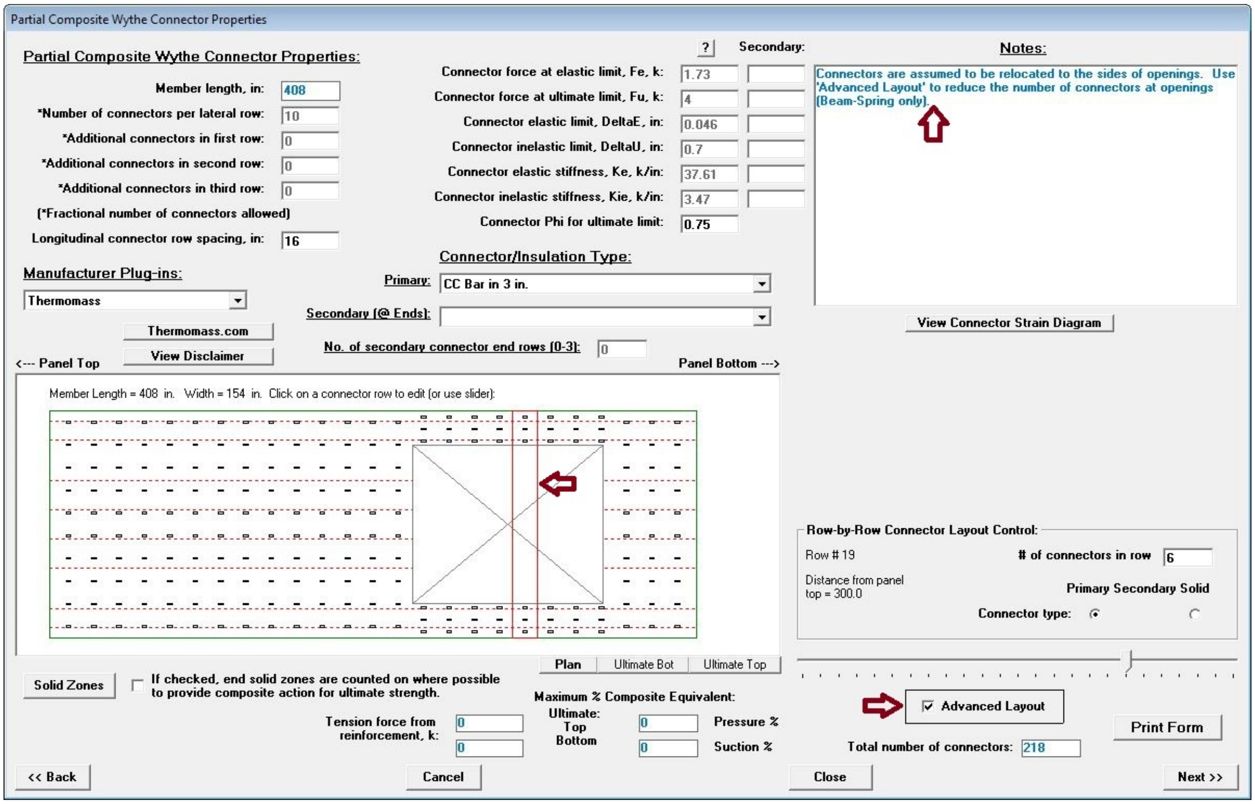
Δ_U = deflection corresponding to the ultimate capacity

Reference: Jaiden Olsen, Salam Al-Rubaye, Taylor Sorensen, Marc Maguire, “DEVELOPING A GENERAL METHODOLOGY FOR EVALUATING COMPOSITE ACTION IN INSULATED WALL PANELS”, Utah State University, 2017. https://digitalcommons.usu.edu/etd/6548/



Advanced Layout:

The 'Advanced Layout' feature on the 'Connectors' screen allows for row-by-row control over wythe connector number and type (available with the Beam-Spring option only). Additional solid zones can also be added. Click on the Plan view to select a row (or use the slider). The '# of Connectors in Row' box is then highlighted for input.



Solid Zone Calculator:

Solid zones are defined as areas at the member ends (and optionally at face lifters) where insulation is held back and the top surface of the bottom wythe concrete is intentionally roughened to provide bond and horizontal shear strength before the top wythe is cast, per ACI 318 16.4.4.2. If used at parapets and/or below grade, solid zones can substantially add to composite action without significantly affecting thermal performance.

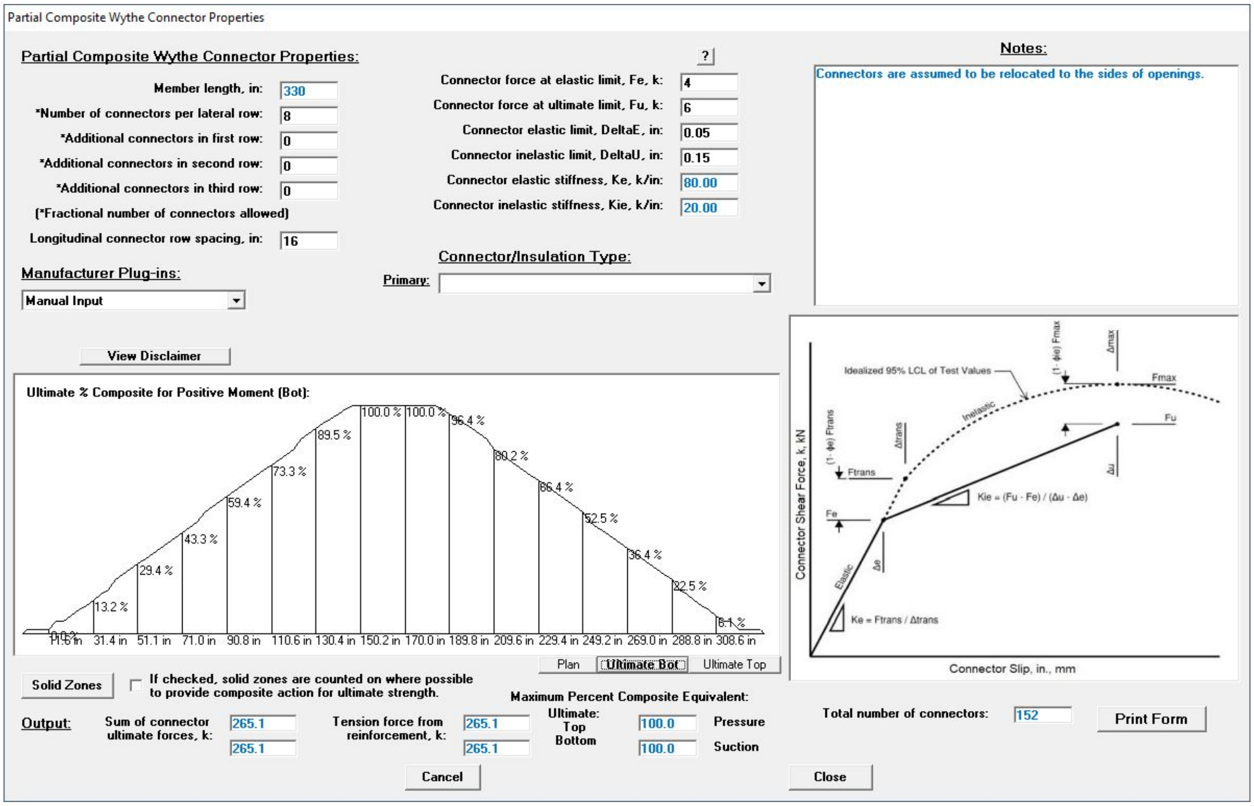
The Solid Zone Calculator requires that all reinforcing and openings be input before using. The calculator compares the solid zone end lengths specified to that required to transfer 100% of the tension reinforcement between wythes. According to ACI 318, the horizontal shear capacity is 80 psi (550 kPa) times the solid area. To this a shear phi-factor of 0.75 is applied, netting 60 psi (410 kPa). The calculator finds the length required at each end to provide 100% composite action for ultimate strength. If satisfied at both ends, then the panel is considered 100% composite for its full length (stresses and deflection will still act as partially composite). If satisfied at one end only, then composite action will peak at that end and slope to zero at the other end.

Solid area that is not sufficient to satisfy ACI is not counted at all for ultimate strength. This is because once the solid bond is fractured, it can no longer contribute to composite action. It can still be used to reduce service stresses, however. As long as the average horizontal shear stress in the solid zone is below 60 psi for the load case under examination, it can be relied on to reduce panel stresses. These values will be displayed in the Notes box on the Beam-Spring Graphs screen.

A note on using solid zones combined with manual input of % composite for stress: It has been found through modeling that there can be a stress reversal at the point the solid zone ends, similar to that due to a fixed-end moment. The LECWall manual input option does not show this. The Beam-Spring method should be selected to account for this effect

Ultimate graphs:

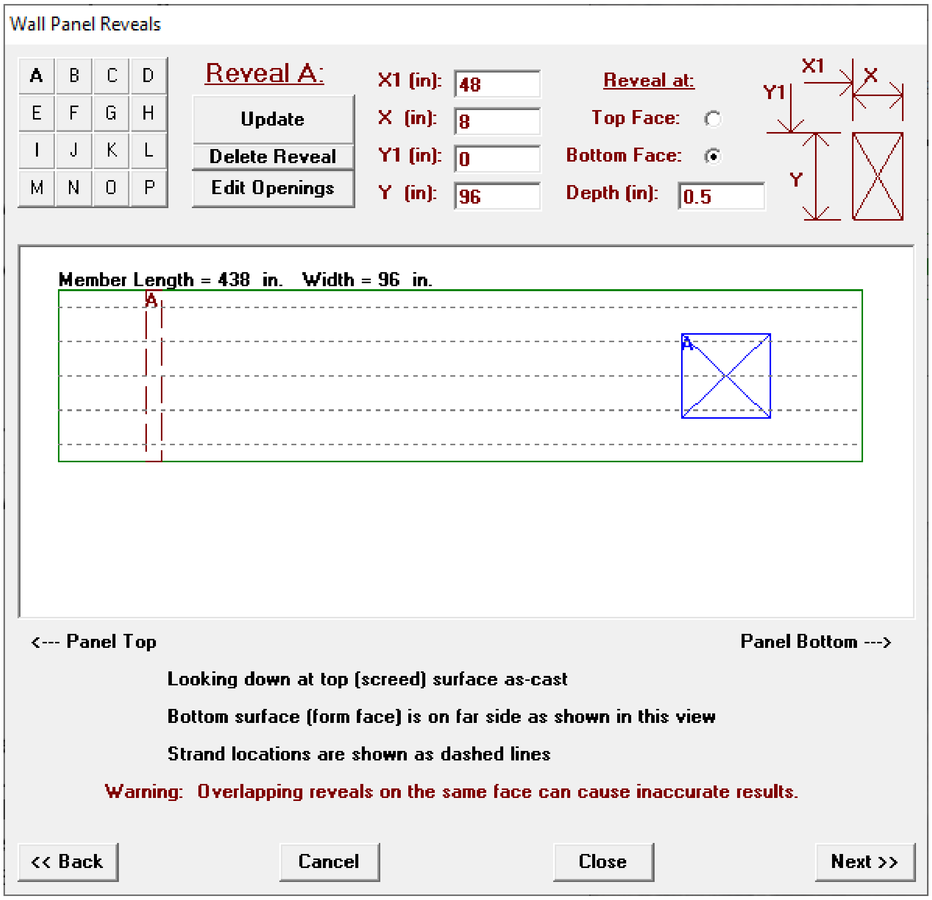
The % Composite for Ultimate graphs determine whether the wythe connectors are sufficient to transfer 100% of the tension reinforcement ultimate strength at points along the panel length. This graph will be in the shape of a pyramid, as there are fewer connectors to contribute as one approaches the panel ends. Therefore, the end connector rows are most important. As such, LECWall offers the option of additional connectors in the first three rows at each end. Some manufacturers also allow the substitution of stiffer connectors in the end rows - these are termed "Secondary Connectors". If solid zones can be counted, the effect of those will be shown in the graph.



Reveals

<Back>

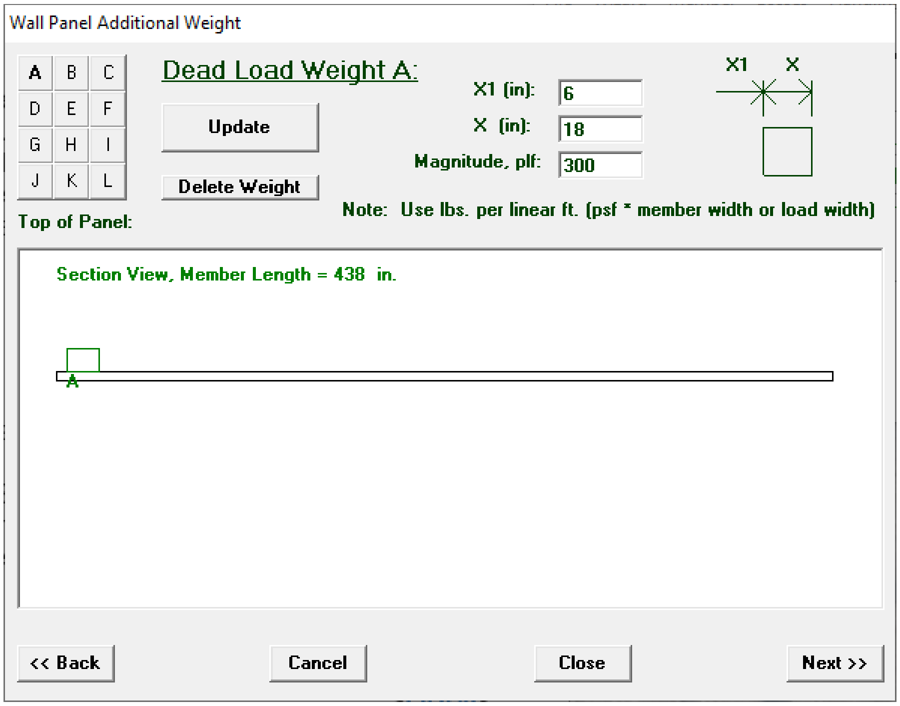
Reveals are depressions in the top or bottom face of the panel which do not penetrate the full thickness of the wythe and do not cut through any reinforcing. This form displays all reveals in plan view in red. Openings are displayed in blue for reference. Up to sixteen reveals can be specified (A-P). **Reveals should not overlap with other reveals or openings.** Reveals can run longitudinally or transverse.



Click the checkbox to specify whether the reveal is on the top or bottom face. Reveals are located relative to the upper left corner of the plan view. Note that the section view is cut looking to the right. The top edge in the plan view is the left edge of the section view. Use the **Update** button to view any changes made to reveals.

Additional Weight

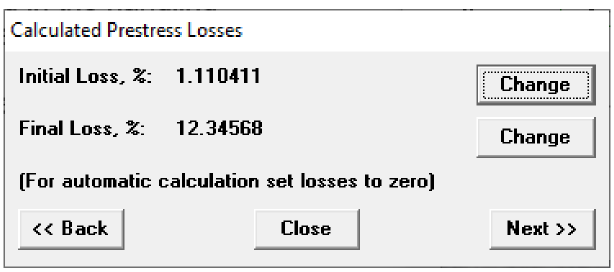
<Back>



This form displays added uniform load in plf or N/m units in edge view in green. It is useful for adding the weight of concrete haunches or any other weight which should be included in the handling calculations. Up to twelve loads can be specified (A-L). Load start is located relative to the left of the edge view (top of panel). X1 is the length of the load along the member. X coordinates run in ascending order from left to right. Use the Update button to view any changes made to loads.

Prestress Losses

<Back>



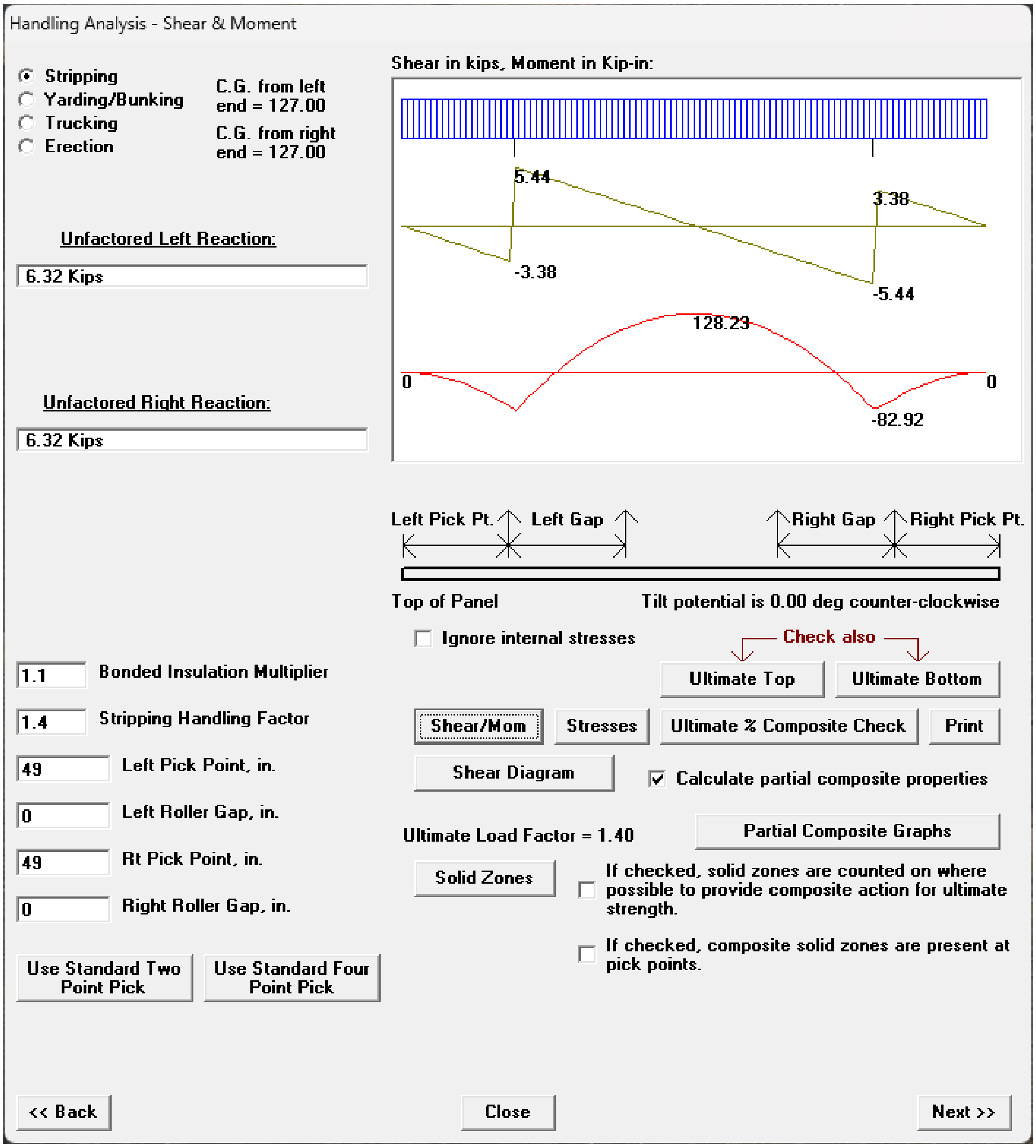
Calculated prestress losses at initial (release) and at final (28 days) are displayed on this form.

Use the **Change** button to manually input a loss percentage. Set the loss percent to zero if you wish it to be calculated automatically. If you manually input a loss percentage, automatic calculation will be turned off on all subsequent runs until the percentage is changed back to zero.

Handling Check

<Back>

The Handling Analysis screen has four parts. The first is a shear and moment display. It is called up by clicking the **Shear/Mom** button after selecting the **Stripping, Yarding, Trucking,** or **Erection** option. The second part is a graphical display of stresses along the member. It is called up by clicking on the **Stresses** button. The third part is an ultimate capacity check of handling forces. Click on **Ultimate Check** to call up this option. Lastly, shear forces can be found by clicking **Shear Diagram**.



Shear and Moment Display:

The panel or column weight is calculated automatically from previous input, multiplied by a user-specified handling factor and used to generate the shear and moment diagrams. At the top of the screen is a representation of the member with its factored weight shown as vertical bars at 100th points. Lifting or bunking points are shown as a vertical black line underneath the member. Below that is the shear diagram, and at the bottom is the moment diagram. Although shear and moment values are calculated at 250 points along the member, the values are an approximation and can vary from an exact solution by one or two percent.

Changing any of the input values on the left of the form, except for % Composite, will change the shear and moment diagrams. Click on **Shear/Mom** to update the diagrams after changing any of these values.

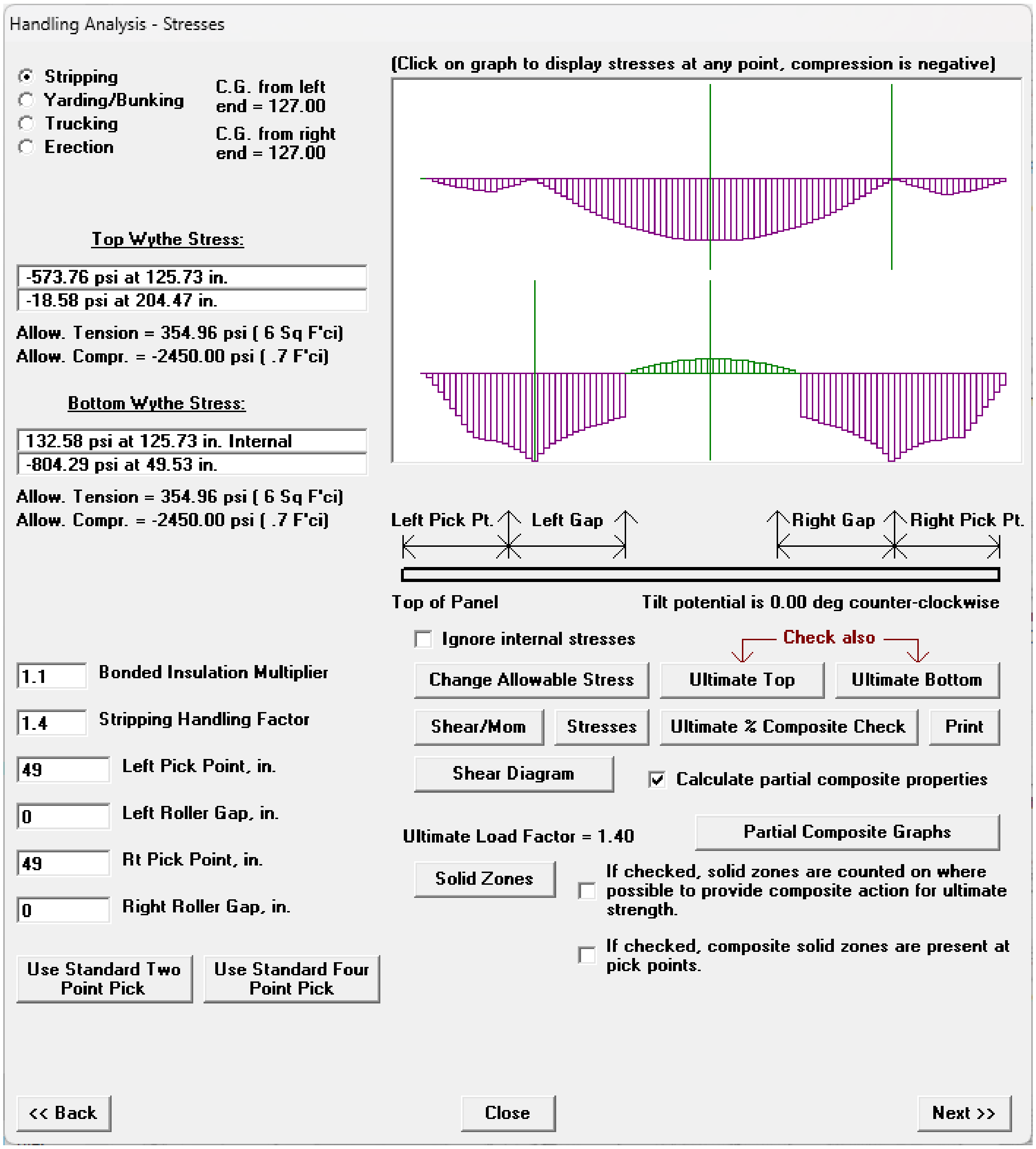
The member can be picked at two, three or four points. Leave the **Left Gap** and **Right Gap** at zero for a two point pick. Add a gap width to the left or right pick point for a three point pick. Add a gap width to both pick points for a four point pick. Cable forces are assumed to be equally balanced by rolling blocks at the two left pick points and the two right pick points. Therefore, the two left reactions will always equal each other and the two right reactions will always equal each other. **Note:** *All lift lines are assumed to* *be vertical. If inclined lines are used, any possible secondary moments induced by them will not be accounted for by the program.* Potential panel tilt is also displayed. Tilt can occur when rollers are used for lifting to equalize the left and right reactions. If all the lift lines are on rollers ending in a single roller/line at top, the member is free to tilt. For erection, it is often necessary for the top of panel tilt up. This would be a clockwise tilt on the LECWall diagram.

Left and right reactions are shown to the left of the shear and moment diagrams. These reactions are based on the member weight only, without any handling factors. They are presented this way since lifting insert capacity charts usually recommend a 4 to 1 factor of safety for tension or 2.67 to 1 for shear.
Note: *For four point picks, as the member is rotated vertical for erection, most of the force from the lower lift line is taken by the insert(s) at the lowest pick point, due to the geometry of the lift lines. The reaction displayed is a total for both lower pick points. Most of this reaction should, therefore, be considered to act at the lowest pick point when selecting the lifting inserts.*

The center of gravity of the member is shown near the top of the window. This can be used to locate proper pick locations.

The "tilt potential" of the member is displayed at the lower right of the window. This value is the estimated member rotation that could result if the member is lifted using roller blocks so that all lifter reactions are equal.

Stress Display:



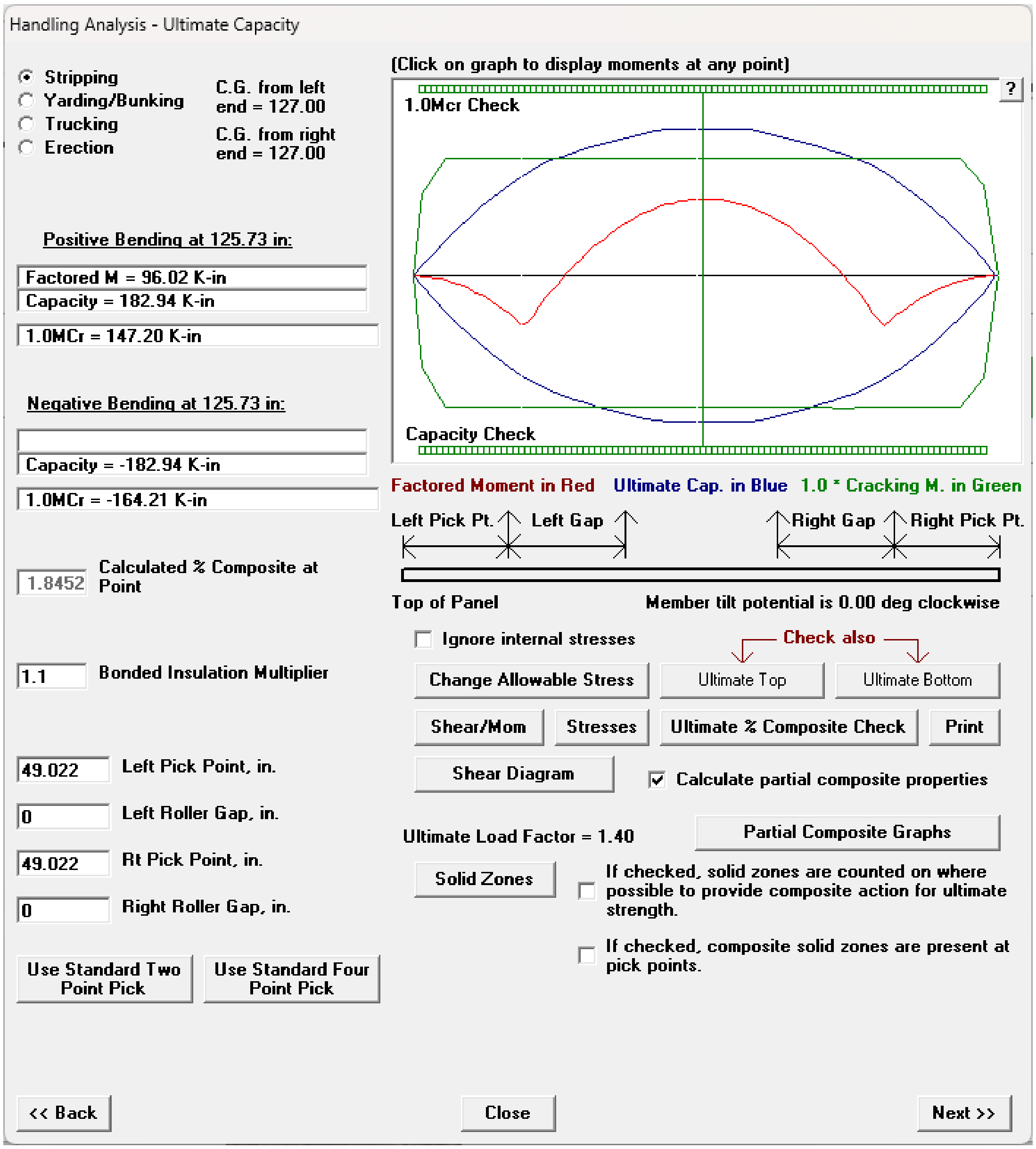
The top graph displays the stresses at the top of the member at 100th points. The bottom graph displays stresses at the bottom of the member. These stresses are adjusted according to the **% Composite** value specified at the Shear and Moment display. Bars below the baseline denote a net compression stress (negative value) and are shown in magenta. Bars above the baseline denote a net tension stress (positive) and are shown in green. Any red colored areas denote where stresses exceed allowable limits. Click on the Change Allowable Stress button to view or change the default stress limits.

Click on the graph at any point to display the stresses at that point numerically. The value will appear in a box to the left of the diagram and a vertical green line will appear on the diagram showing where the stresses occur. Up to two points can be selected and displayed simultaneously. These points are stored in the problem file so that critical locations need not be reentered each time the file is run.

Note that LECWall includes the non-structural wythe in the section modulus calculation for zero percent composite. For example, with two wythes, one 96” by 3” and the other 96” by 4”, non-composite I would be 512in4 + 216in4 = 728 in4. Snc = I/c = 728/2” = 364 in3, vs 256in3 for the 4” main structural wythe alone.

The program will automatically show the internal tension stress at the insulation face if it is greater than the tension stress at the outer face. When this occurs, the wythe stress readout will be marked "Internal".

Check of Ultimate Capacity for Handling Forces:



Statically calculated unfactored moments are multiplied by the ultimate load factor **instead** of the user-specified handling (or impact) factor. (This ultimate load factor is initially set to 1.4, but can be changed by selecting Defaults from the main pull-down menu and clicking the **Allowable Stresses** button.) The factored handling moment is displayed in red. Values above the baseline denote positive moment (or simple span bending) and values below the baseline denote negative moment (or cantilever bending).

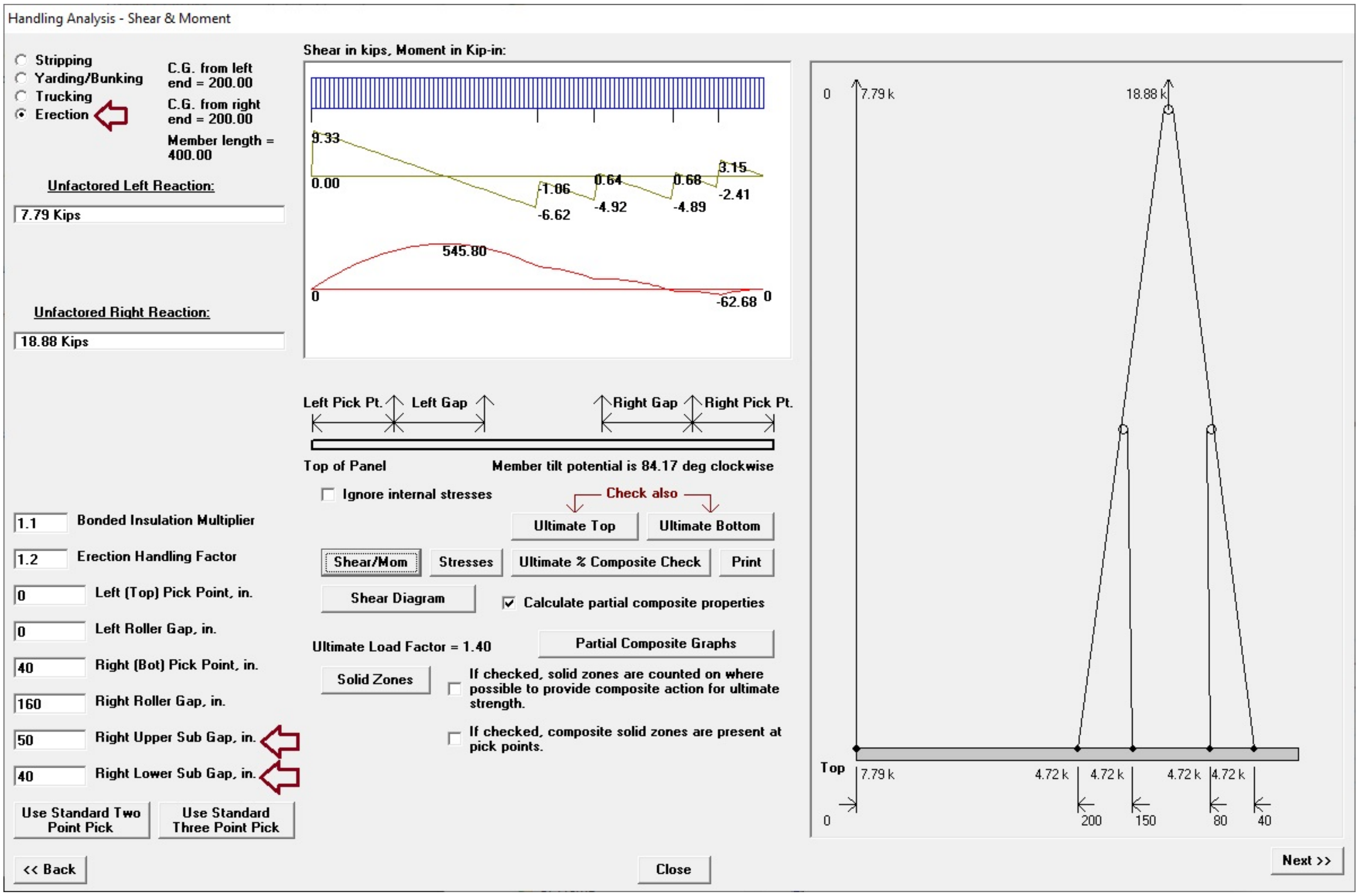
Strain compatibility is used to determine the member's ultimate capacity in the horizontal position for both positive and negative moment. The moment capacity envelope is shown in blue. The handling moment (red line) should be completely contained by the capacity envelope (blue line), otherwise capacity has been exceeded.

Note that zero percent composite for ultimate strength uses the reinforcing and compression block of the main structural wythe only and ignores any contribution from the other wythe.

1.0 (or 1.2) times the theoretical cracking moment can also be displayed (per ACI 318). See Methodology for further information.

Click on the graph at any point to display moments at that point. Select a point where the blue and red lines are closest as the critical point. This point is stored in the problem file so that critical locations need not be reentered each time the file is run.

Optional Sub-Gaps for Erection Handling <Back>



Occasionally, additional rollers are desired for erection in order to keep stresses down. These can be input directly as shown by specifying one or two "sub gaps" within the right (lower) roller gap (see example above). Note that LECWall always assumes two lines of support to maintain static determinacy.

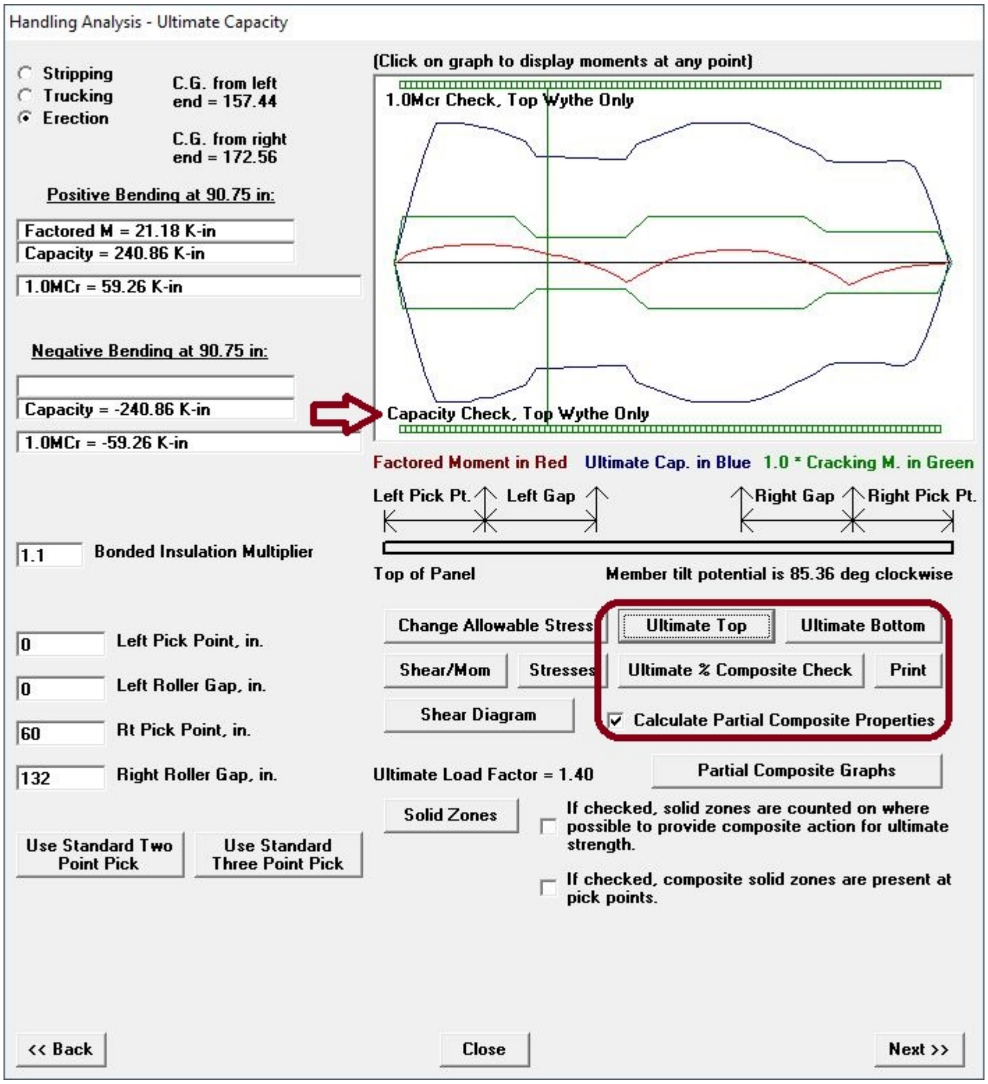
If an erection lower line pick point is above the panel center of gravity, a warning will be displayed. This warning is based on the PCI Design Handbook Fig. 8.6.1f. The concern is that the panel could become unstable or oscillate as it is pulled vertical, One way to avoid this is to pull the lower roller block toward the base of panel with an added guide line, keeping the roller past the center of gravity as the panel is tilted vertical.

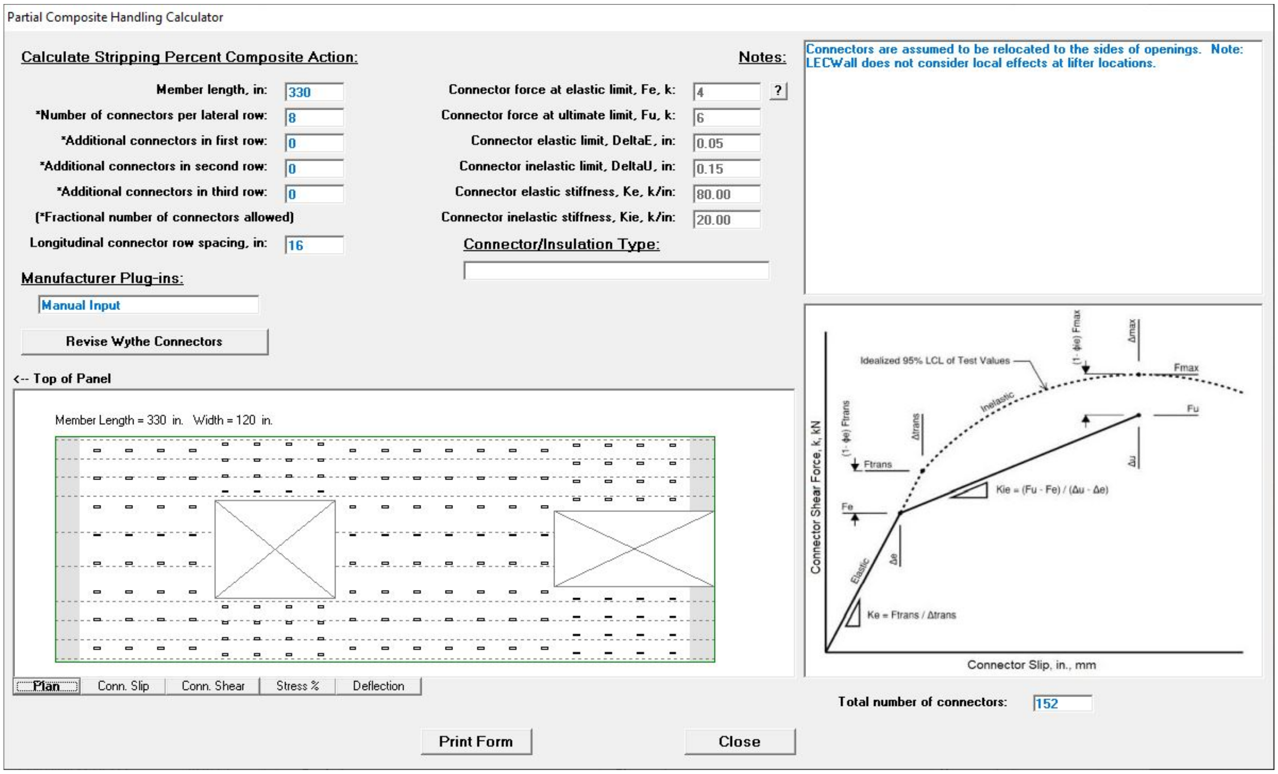
Partial Composite Handling <Back>

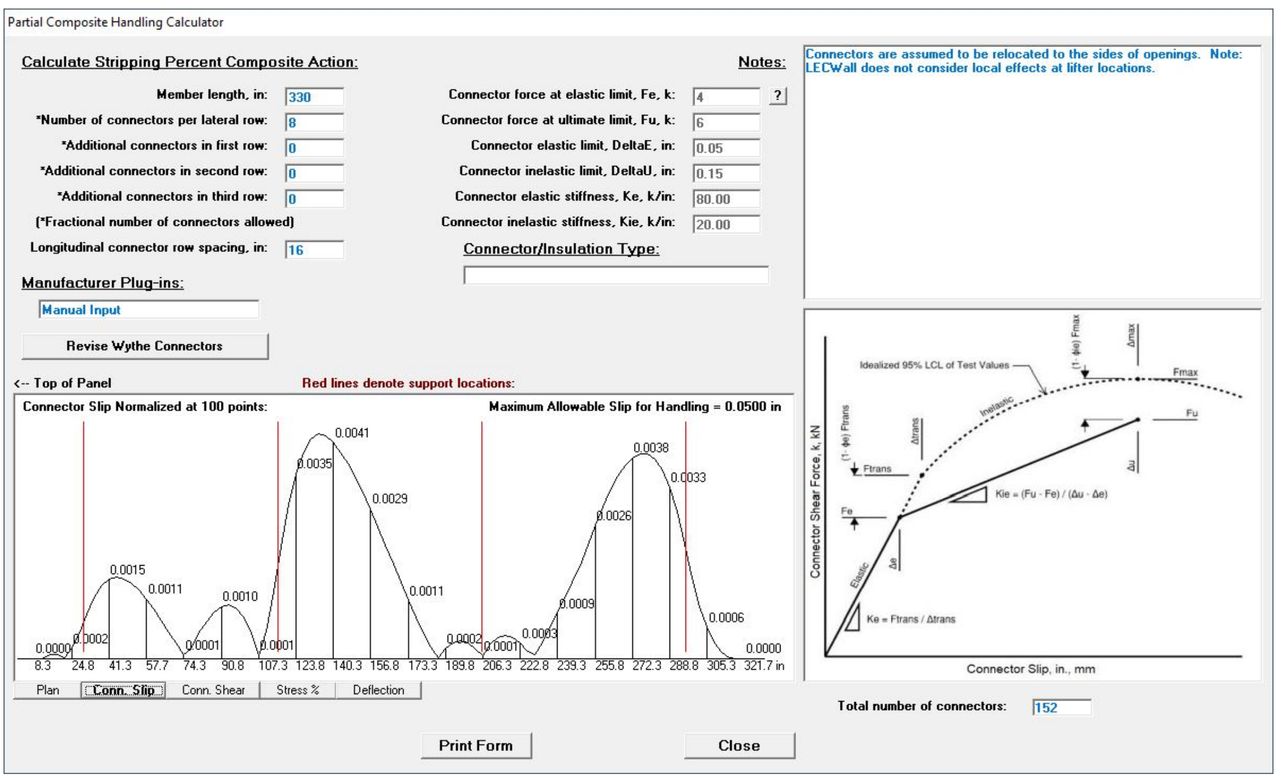
Partial composite properties for handling can be calculated automatically if wythe connector parameters have been specified on the Connectors screen. To use, select "Calculate Partial Composite Properties" from the Handling screen. Click on "Partial Composite Graphs" to view connector shear and slip.

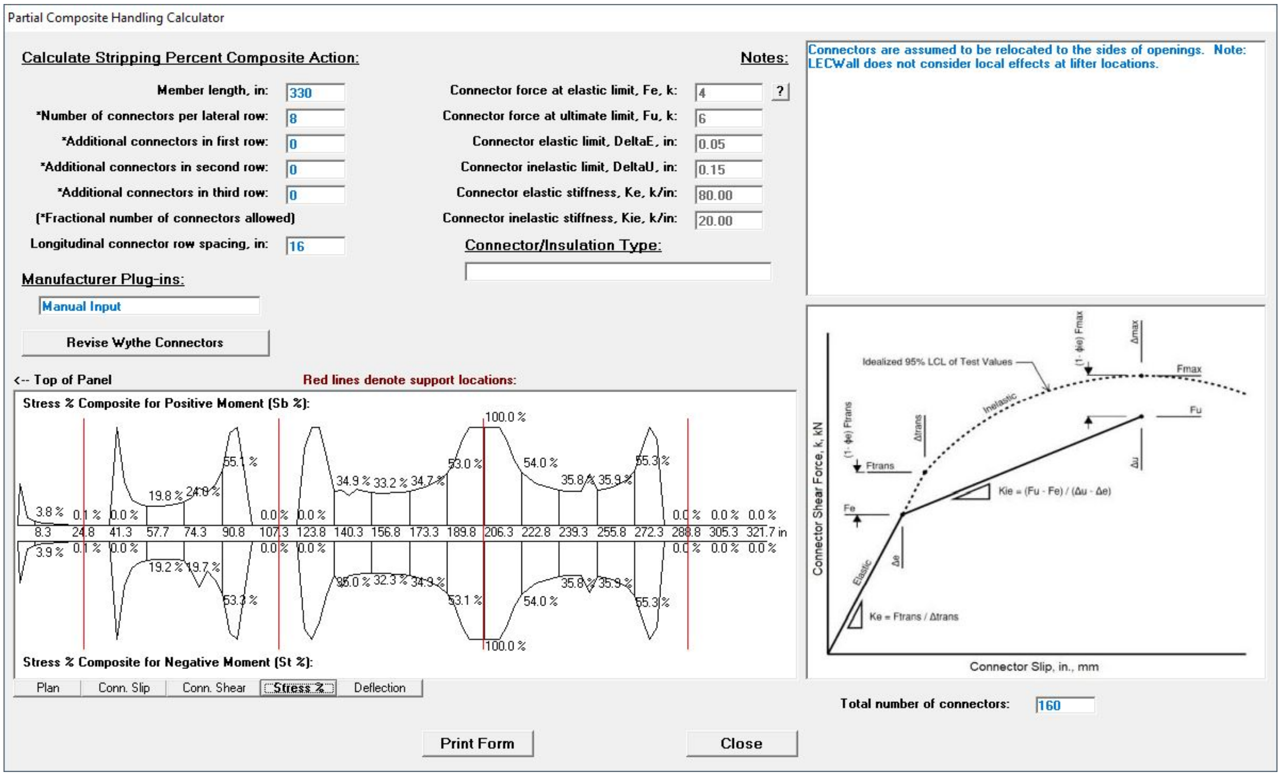
The Bonded Insulation Multiplier accounts for the increase in composite action when the insulation is not debonded from the concrete wythes. This adds from 10% to 40% added connector shear capacity, depending on the manufacturer. Since insulation bond can break down over time, it's allowed for handling but not in-place loads. This helps reduce stresses when handling, due to the greater composite action.

The **Ultimate % Composite Check** uses the % composite provided by the connectors to estimate the % of reinforcing engaged between the wythes at each point along the member. This is not as accurate as the individual wythe checks (**Ultimate Top** and **Ultimate Bottom**) available with the Beam-Spring method. This is because there can be a shear reversal at some lift points which could theoretically limit the effectiveness of the connectors at that point. Therefore, the individual wythe checks would govern over the % Composite check for ultimate capacity.




Partial Composite Handling Calculator
Calculate Stripping Percent Composite Action:
Member length, in: 330
*Number of connectors per lateral row: 8
*Additional connectors in first row: 0
*Additional connectors in second row: 0
*Additional connectors in third row: 0
(*Fractional number of connectors allowed)
Longitudinal connector row spacing, in: 16
Connector force at elastic limit, Fe, k: 4
Connector force at ultimate limit, Fu, k: 6
Connector elastic limit, DeltaE, in: 0.05
Connector inelastic limit, DeltaU, in: 0.15
Connector elastic stiffness, Ke, k/in: 80.00
Connector inelastic stiffness, Kie, k/in: 20.00
Connector/Insulation Type:
Notes:
Connectors are assumed to be relocated to the sides of openings. Note: LECWall does not consider local effects at lifter locations.
Manufacturer Plug-ins:
Manual Input
Revise Wythe Connectors
<-- Top of Panel
Member Length = 330 in. Width = 120 in.
Plan
Conn. Slip
Conn. Shear
Stress %
Deflection
Idealized 95% LCL of Test Values
Fmax
Fu
Ftrans
Fe
Kie = (Fu - Fe) / (Δu - Δe)
Ke = Ftrans / Δtrans
Connector Shear Force, k, kN
Connector Slip, in., mm
Total number of connectors: 152
Print Form
Close


Partial Composite Handling Calculator
Calculate Stripping Percent Composite Action:
Member length, in: 330
*Number of connectors per lateral row: 8
*Additional connectors in first row: 0
*Additional connectors in second row: 0
*Additional connectors in third row: 0
(*Fractional number of connectors allowed)
Longitudinal connector row spacing, in: 16
Connector force at elastic limit, Fe, k: 4
Connector force at ultimate limit, Fu, k: 6
Connector elastic limit, DeltaE, in: 0.05
Connector inelastic limit, DeltaU, in: 0.15
Connector elastic stiffness, Ke, k/in: 80.00
Connector inelastic stiffness, Kie, k/in: 20.00
Connector/Insulation Type:
Notes:
Connectors are assumed to be relocated to the sides of openings. Note: LECWall does not consider local effects at lifter locations.
Manufacturer Plug-ins:
Manual Input
Revise Wythe Connectors
<-- Top of Panel
Red lines denote support locations:
Connector Slip Normalized at 100 points:
Maximum Allowable Slip for Handling = 0.0500 in
0.0000 0.0002 0.0015 0.0011 0.0001 0.0010 0.0001 0.0035 0.0041 0.0029 0.0011 0.0002 0.0001 0.0003 0.0009 0.0026 0.0038 0.0033 0.0006 0.0000
8.3 24.8 41.3 57.7 74.3 90.8 107.3 123.8 140.3 156.8 173.3 189.8 206.3 222.8 239.3 255.8 272.3 288.8 305.3 321.7 in
Plan
Conn. Slip
Conn. Shear
Stress %
Deflection
Idealized 95% LCL of Test Values
Fmax
Fu
Ftrans
Fe
Kie = (Fu - Fe) / (Δu - Δe)
Ke = Ftrans / Δtrans
Connector Shear Force, k, kN
Connector Slip, in., mm
Total number of connectors: 152
Print Form
Close



Handling Print Options:

Select the **Print** button to call up the **Print Selected Items** form. Click on the **Print Input Data** check box to send the input data for the current file to the printer. The check boxes for three handling conditions are checked by default. Un-check the box for any condition you do not want to print. Click on the **Print Handling Shear and Moment Diagrams** check box to print the shear and moment diagrams for the selected handling conditions. Click on **Print Handling Shear Graphs** to print the shear graphs (useful for checking the wythe connectors). Click on **Print Handling Ultimate Capacity Graphs** to print ultimate capacity graphs. Click on **Print Handling Stress Graphs** to print the stress graphs.

Select **Done** to close the Handling form and return to the main menu.

Change Allowable Stress:

Allowable tension and compression stress coefficients are displayed on this form, which is displayed by clicking the **Change Allowable Stress** button on the Handling Check - Stresses form. The compression coefficient is multiplied by lambda and F'c or F'ci. The tension coefficient is multiplied by lambda and the square root of F'c or F'ci. The default values are as recommended by the PCI Design Handbook, Table 5.2.2.

Use the **Change** button to manually input a stress coefficient. Set the coefficient to zero if you wish to use the default values.

If you manually input a coefficient, the default values will be turned off on all subsequent runs until the coefficient is changed back to zero.

Percent Partial Composite

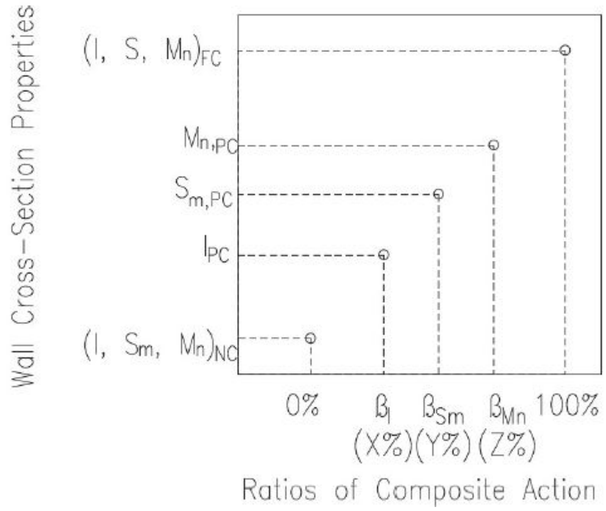
<Back>

The program determines the member's composite properties by the following method:

Section properties are calculated at 100th points assuming full 100% composite action, then again, assuming no (0%) composite action. The partially composite properties are based on the percent difference between full composite and non-composite area, moment of inertia and section modulus of the section. For example, 80% composite moment of inertia ("I_{PC}") would be:

$$I_{PC} = I(0\%) + (I(100\%) - I(0\%)) * 0.80$$

The program makes no assumptions about what percent composite action, if any, can be counted on for a particular member type. It is up to the engineer/user to provide reasonable values based on calculation, load tests and experience, noting that insulation bond can break down over time with a resultant loss of composite action.



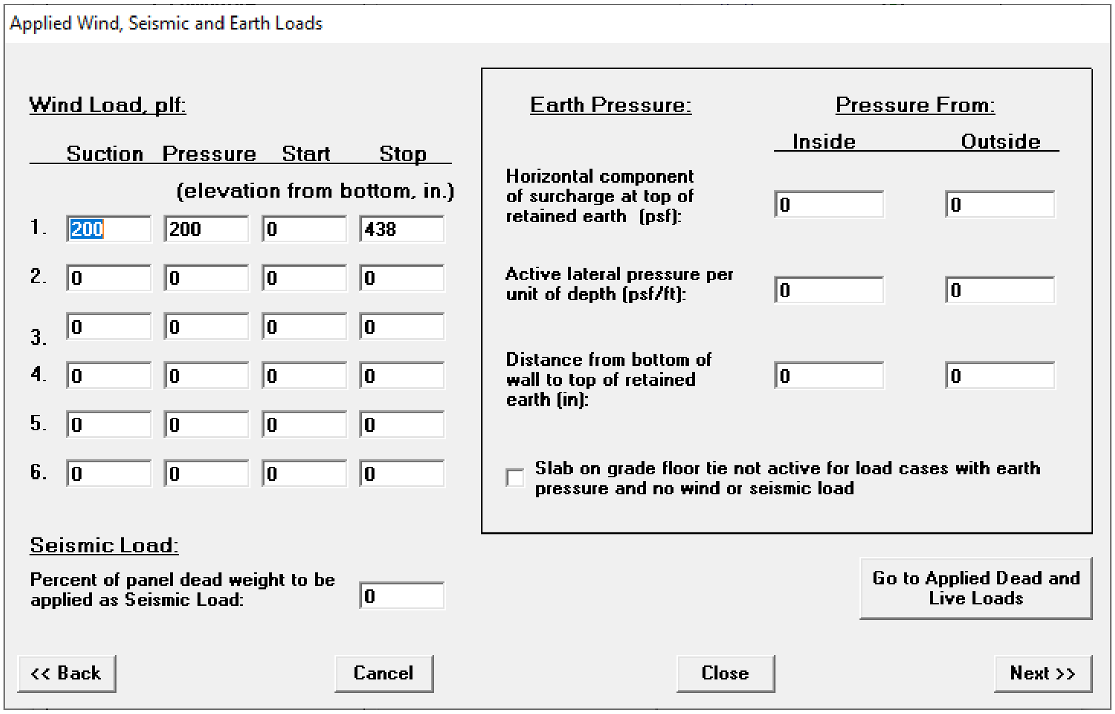
Here are some references regarding insulated panel composite action:

- Wade, T. G., Porter, M. L., and Jacobs, D. R., "Glass-Fiber Composite Connectors for Insulated Concrete Sandwich Walls," *Report*, Engineering Research Institute, Iowa State University, Ames, 1988.
- Einea, Salmon, Fogarasi, Culp, and Tadros, "State-of-the-Art of Precast Concrete Sandwich Panels," *PCI Journal*, V. 36, No. 6, Nov-Dec 1991, pp 78-98.
- Bush, T. D., Jr. and Stine, G. L., "Flexural Behavior of Composite Precast Concrete Sandwich Panels with Continuous Truss Connectors," *PCI Journal*, V. 39, No. 2, Mar-Apr 1994, pp 112-121
- Jagdish C. Nijhawan, "Insulated Wall Panels - Interface Shear Transfer," *PCI Journal*, V. 43, No. 3, May-Jun 1998, pp 98-101

Applied Loads:

<Back>

Click on the **Applied Loads** button to specify the wind and gravity loads to be resisted by the member. **Wind Forces** are in units of lbs. per lineal ft. (or Newtons per Meter) and can act in pressure or suction.

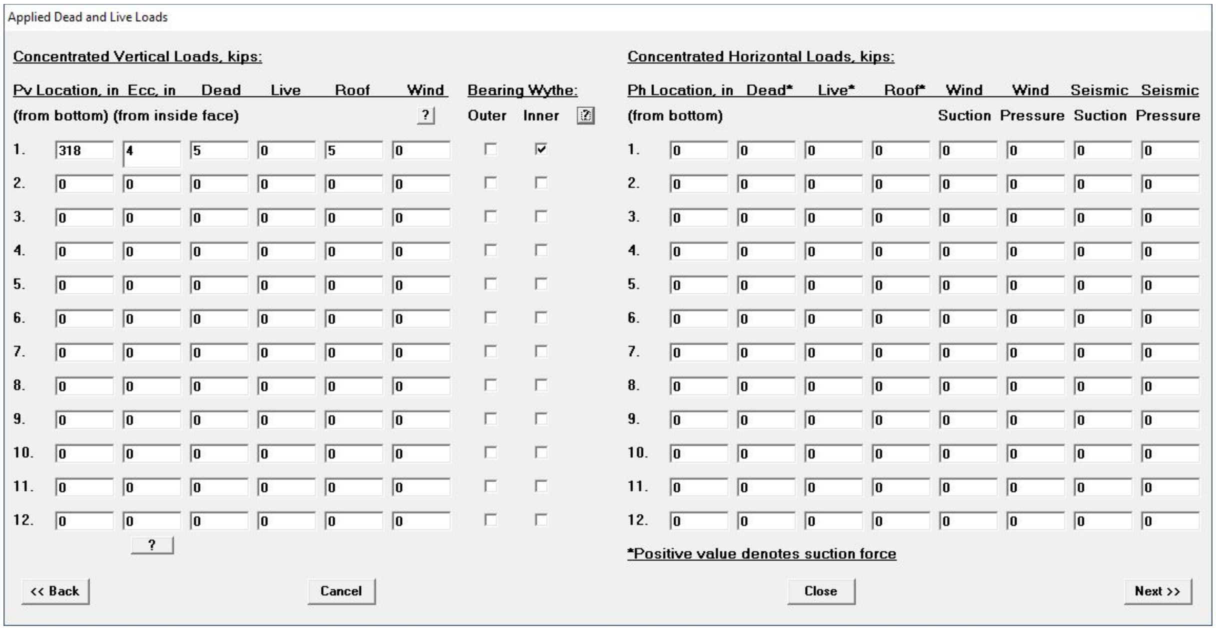


Earth Pressure can be applied to the panel, with or without a surcharge. Note that the surcharge and lateral pressure are in lbs. per *square* ft. (or kPa) and that the surcharge requested is the *horizontal* component only. (The horizontal surcharge component can be found by multiplying the vertical surcharge by the appropriate active soil pressure coefficient.) The program multiplies the load values by the width of the panel before applying the load. Both outside and inside pressures should always be positive, acting toward the wall. Input the top of retained earth level from base of member as requested.

Floor tie not active when lateral earth pressure applied: Check this box if backfill is to be applied before the floor tie is connected. The stress and member bow due to this operation will be calculated without the benefit of the floor tie. **Note that the surcharge is also applied before the floor tie is active. If the surcharge is not to be applied until after occupancy, then a separate run could be made to check the panel with the surcharge and floor tie active.** All other loads will still be applied assuming the floor tie becomes active.

Seismic load is applied as a percentage of the panel weight for out-of-plane analysis of vertical spans. Note that LECWall looks at the panel behavior as a whole, so it won't check for local or internal effects, such as tension on the wythe connectors. One can use ASCE 7-16, 12.11.1 to find the seismic force generated by the outer wythe and check it against the connector capacity in shear and tension. The seismic force is the larger of (a) 0.4SDS*Ie*Wp, or (b) 0.1 Wp.

Wp – wall weight
Ie – importance factor in accordance with 11.5.1 of ASCE 7
SDS – design, 5%-damped spectral response acceleration parameter at short periods in accordance with 11.4.5 of ASCE 7.


Applied Dead and Live Loads

Concentrated Vertical Loads, kips:

	Pv Location, in (from bottom)	Ecc, in (from inside face)	Dead	Live	Roof	Wind	Bearing Wythe: Outer	Inner
1.	318	4	5	0	5	0	☐	☑
2.	0	0	0	0	0	0	☐	☐
3.	0	0	0	0	0	0	☐	☐
4.	0	0	0	0	0	0	☐	☐
5.	0	0	0	0	0	0	☐	☐
6.	0	0	0	0	0	0	☐	☐
7.	0	0	0	0	0	0	☐	☐
8.	0	0	0	0	0	0	☐	☐
9.	0	0	0	0	0	0	☐	☐
10.	0	0	0	0	0	0	☐	☐
11.	0	0	0	0	0	0	☐	☐
12.	0	0	0	0	0	0	☐	☐

Concentrated Horizontal Loads, kips:

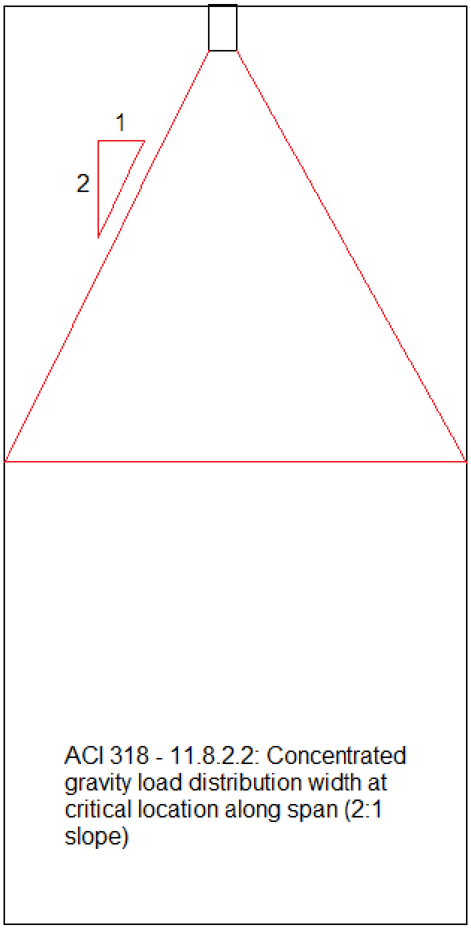
	Ph Location, in (from bottom)	Dead*	Live*	Roof*	Wind Suction	Wind Pressure	Seismic Suction	Seismic Pressure
1.	0	0	0	0	0	0	0	0
2.	0	0	0	0	0	0	0	0
3.	0	0	0	0	0	0	0	0
4.	0	0	0	0	0	0	0	0
5.	0	0	0	0	0	0	0	0
6.	0	0	0	0	0	0	0	0
7.	0	0	0	0	0	0	0	0
8.	0	0	0	0	0	0	0	0
9.	0	0	0	0	0	0	0	0
10.	0	0	0	0	0	0	0	0
11.	0	0	0	0	0	0	0	0
12.	0	0	0	0	0	0	0	0

*Positive value denotes suction force

<< Back | Cancel | Close | Next >>

Concentrated Loads: Input **Superimposed Dead Load** only, not panel self-weight. For non-bearing members this value would usually be zero. Load eccentricity is specified in relation to the inside face of the member. If the load is applied to the top of the member or in a pocket, then the eccentricity would be a negative number. For example, if the member is 8 in. thick and the load is centered on the top of the panel, the eccentricity would be -4 in. If the load is on a haunch extending 6 in. from the inside face, the eccentricity would be 6 in. If stems are on the inside, then eccentricity is measured from the face of the stems, not the Inside face of wall. The same applies for the other vertical loads. Note that horizontal **Wind Suction** and **Pressure** are applied separately and not combined. The same applies to horizontal **Seismic** loads. For the Beam-Spring option only, it is possible to select the bearing wythe. Select both or neither for bearing on both wythes, such as with a steel tube haunch anchored through both wythes, with tension possible.

Concentrated gravity load distribution: Per ACI 318 - 11.8.2.2, concentrated vertical loads are assumed to be distributed at a slope of 2 vertical to 1 horizontal. If the load width at the critical location along the vertical span is less than the full width of panel, then only the load distribution width can be considered. This may require narrowing the panel width in LECWall, as LECWall cannot check this criteria automatically.



Support Locations:

<Back>

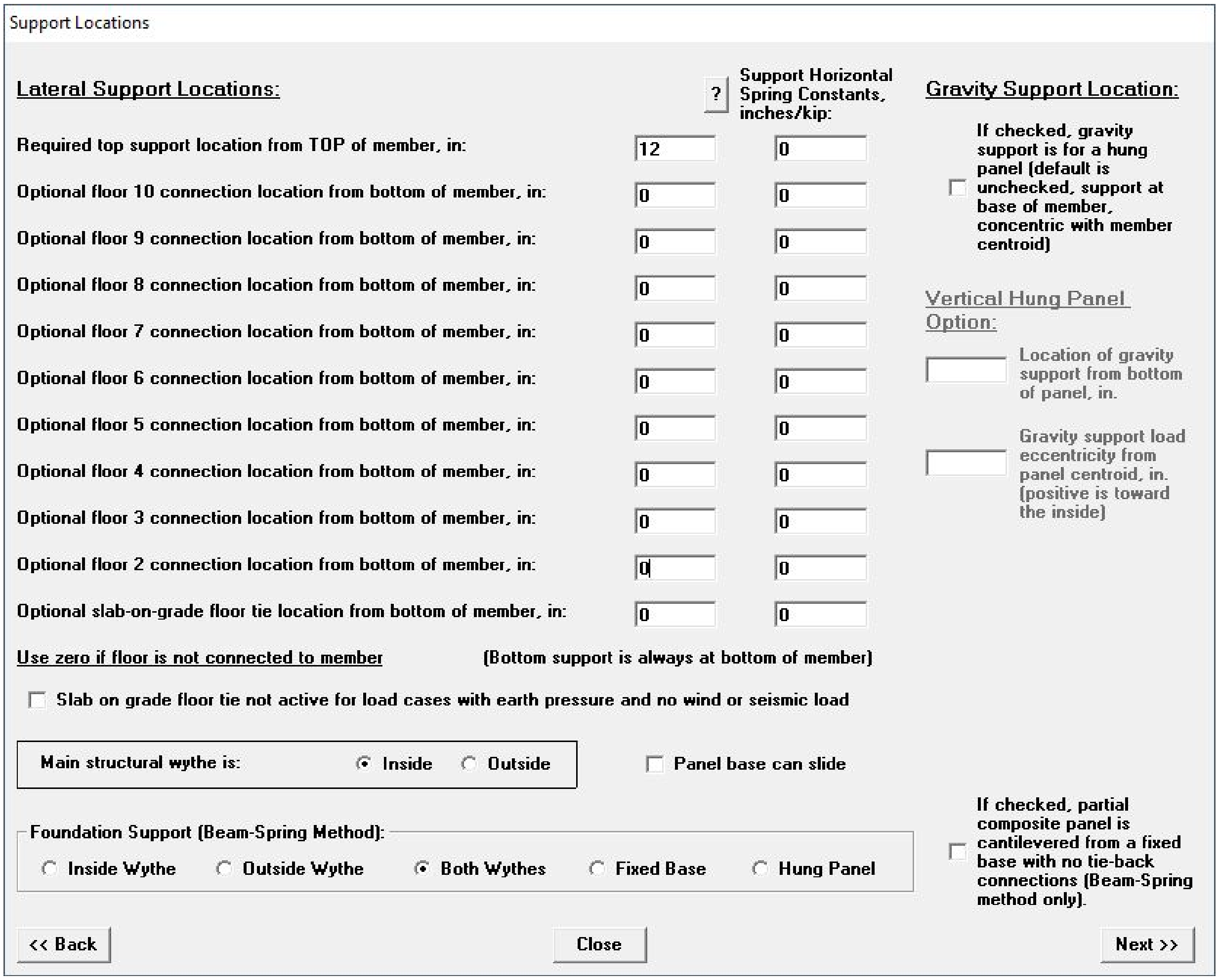
Click on the **Supports** button to specify the support locations and locate the **main structural wythe** of multi-wythe members.

All superimposed loads are assumed to act at the location of the support. If a beam bears in an 18 in. deep pocket at the top of the member, the top support location would be 18 in. - measured from the **top** of the member.

If the member is to be tied into the first floor slab or a mezzanine at a point **above** the base of the member, its location should be entered as the distance of the connection from the **bottom** of the member. Input zero if the floor is not connected to the member or if the floor connection is at the bottom of the panel. See the Methodology section for recommendations for the design of the floor connection.

The main structural wythe was specified in the Section Dimensions window as either top or bottom. Here it needs to be specified as **inside** or **outside**. Wind pressure acts inward, toward the inside of the member, while suction acts outward. For wall panels, the main structural wythe is usually on the inside, to better carry any applied loads from the inside structure of the building.

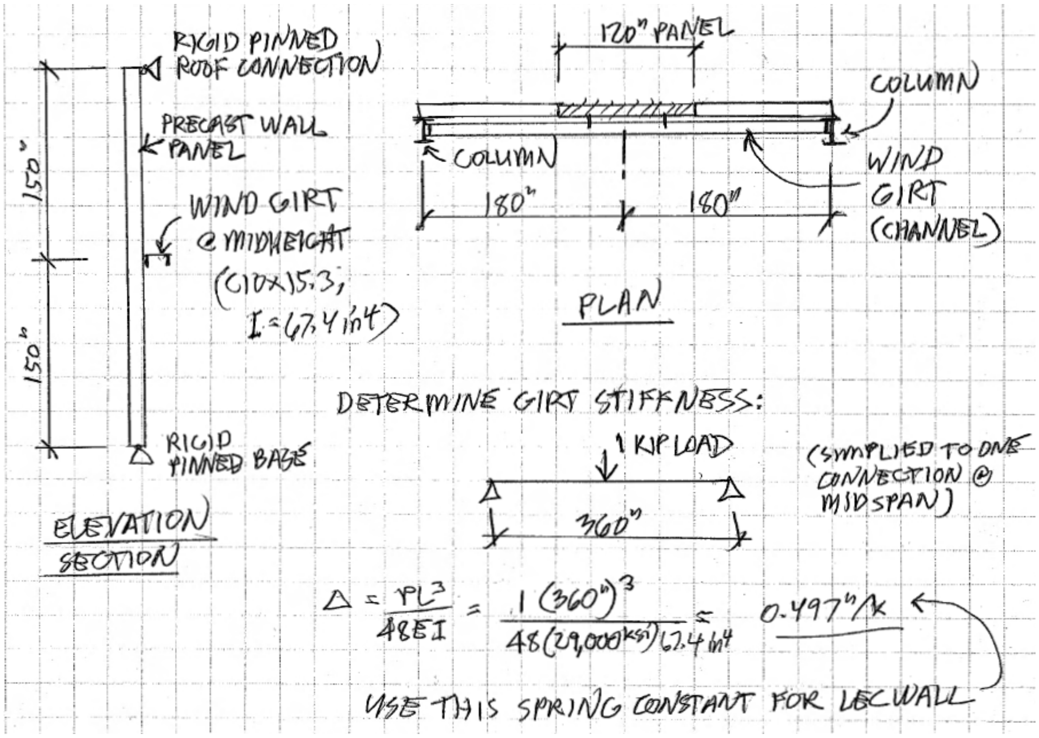
Foundation Support (Beam-Spring only): Select the wythe that will be supported at the base of panel. This is usually either Both Wythes or the Main Structural Wythe, which is usually the Inside Wythe. There is an option for a **sliding panel base**. This can be used for a stacked load-bearing panel with P-Delta effects where the base of panel is not at a lateral support (Beam-Spring only). At least two lateral supports are needed for stability.



Spring Constants: <Back>

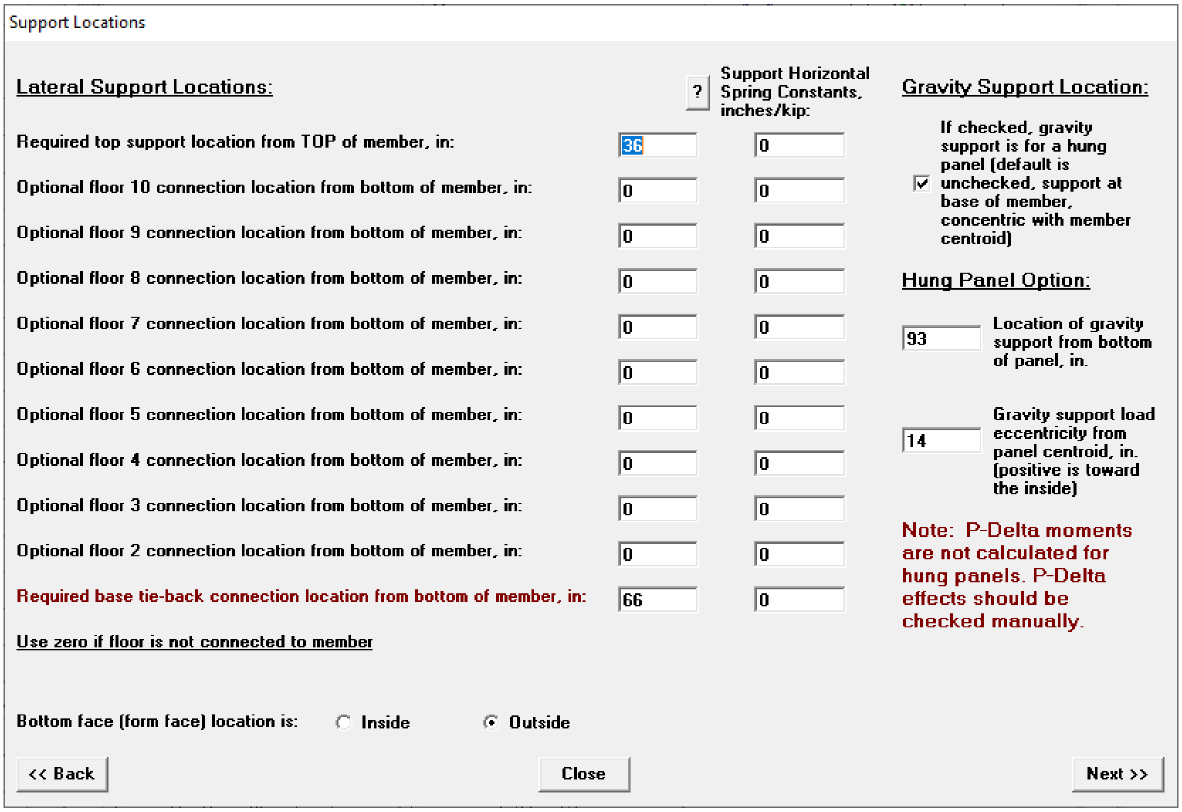
Spring supports are useful for modeling ductile or flexible connections. If restraint tieback force due to temperature or other loads becomes too large, a ductile restraint can be used to reduce the overstress in the connection. Another use is to model the effect of a flexible wind girt (see example below). Use a Spring Constant of 0 (zero) for a rigid support (default). Flexible supports are input as the horizontal deflection per kip of load, either in inches per kip or mm per kN. This value will usually be very small. Larger values increase support flexibility.

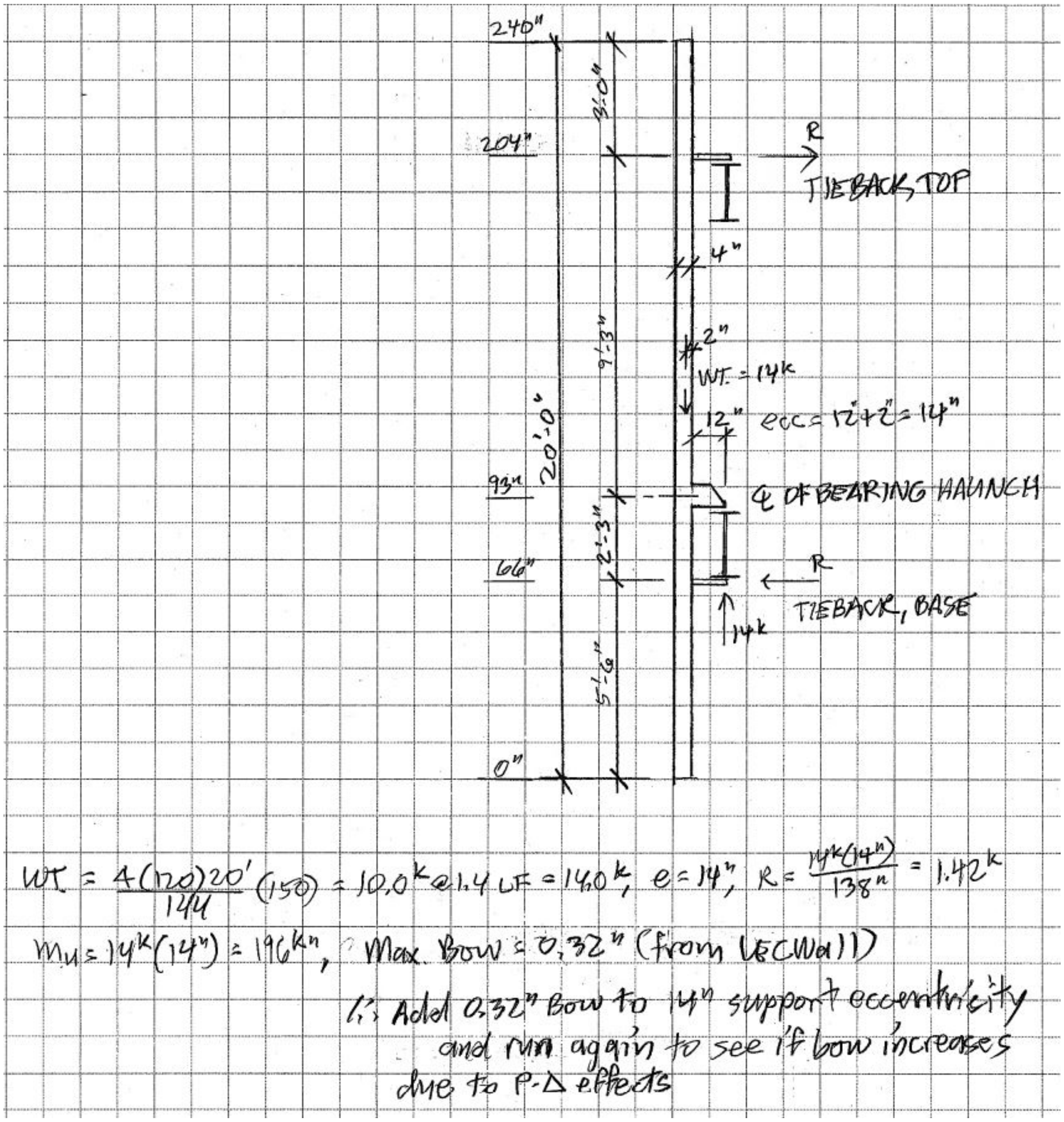
To specify horizontal spring constants at supports, select **Capacity** from the main pull-down menu then click on the **Supports** button at the upper left.



Hung Panels: <Back>

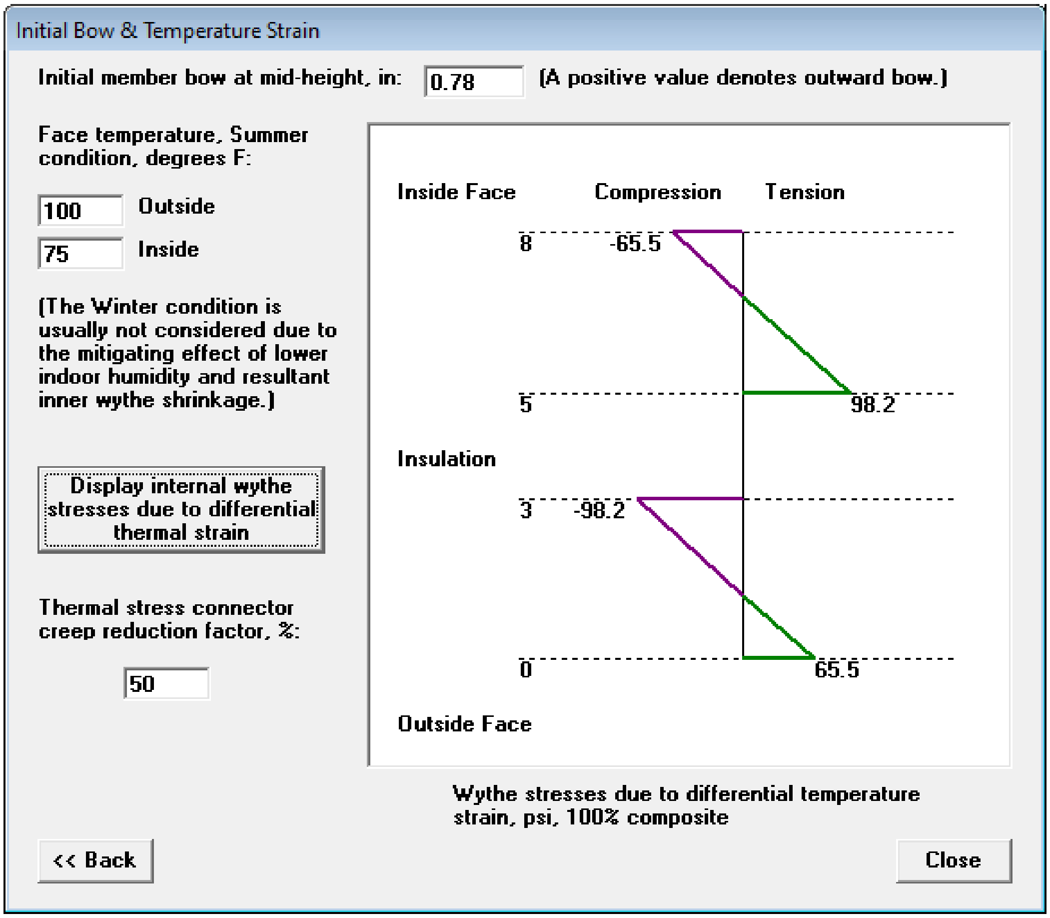
In some cases the panel may not bear at the foundation base, but sit eccentrically on a support haunch or beam. If so, check the option for a Hung Gravity Support. An example is shown below. The gravity bearing may not be at the same elevation as the base tie-back connection. The tie-back elevation is input at lower left (see note in red). **Note that the gravity support load eccentricity is measured from the panel *centroid*, not the inside face, as is typical elsewhere.**


Support Locations
Lateral Support Locations:
?
Support Horizontal Spring Constants, inches/kip:
Gravity Support Location:
Required top support location from TOP of member, in:
36
0
Optional floor 10 connection location from bottom of member, in:
0
0
Optional floor 9 connection location from bottom of member, in:
0
0
Optional floor 8 connection location from bottom of member, in:
0
0
Optional floor 7 connection location from bottom of member, in:
0
0
Optional floor 6 connection location from bottom of member, in:
0
0
Optional floor 5 connection location from bottom of member, in:
0
0
Optional floor 4 connection location from bottom of member, in:
0
0
Optional floor 3 connection location from bottom of member, in:
0
0
Optional floor 2 connection location from bottom of member, in:
0
0
Required base tie-back connection location from bottom of member, in:
66
0
Use zero if floor is not connected to member
If checked, gravity support is for a hung panel (default is unchecked, support at base of member, concentric with member centroid)
Hung Panel Option:
93
Location of gravity support from bottom of panel, in.
14
Gravity support load eccentricity from panel centroid, in. (positive is toward the inside)
Note: P-Delta moments are not calculated for hung panels. P-Delta effects should be checked manually.
Bottom face (form face) location is:
Inside
Outside
<< Back
Close
Next >>



Initial Bow & Temperature Strain: <Back>

Long and slender members such as wall panels usually have an initial bow due to differential shrinkage, non-concentric prestress, and/or storage in a deflected position. The value to input for initial bow should be based on the performance history of the member type, its degree of slenderness and the engineer's best judgement. The initial bow is combined with the bow due to temperature strain, applied loads and wind loads and is used for an iterative P-Delta analysis. The total bow, as magnified by the P-Delta analysis, is displayed in the **Output** window for both wind pressure and suction for a given load case. See the Methodology section for further information.



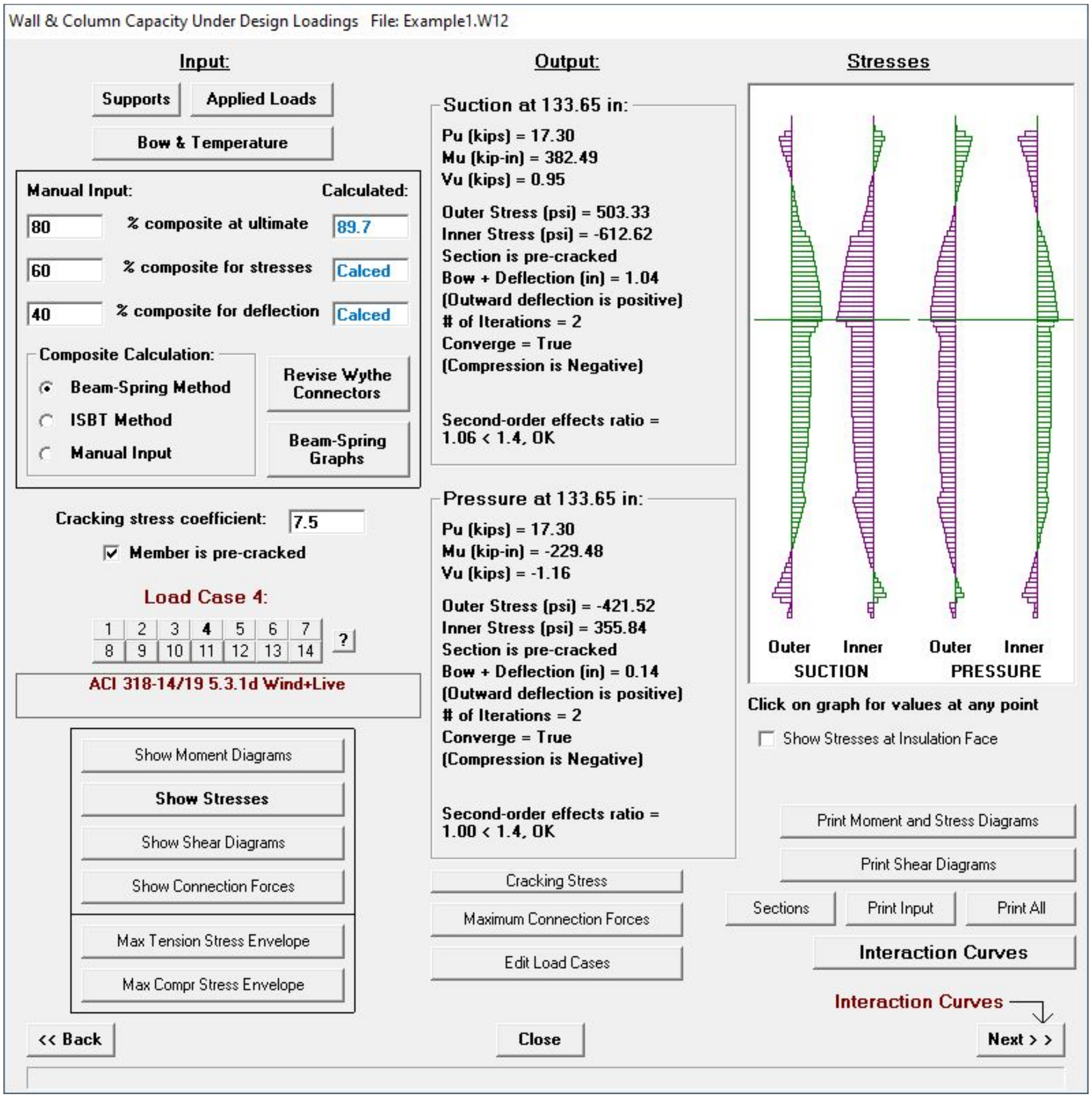
Bow due to differential temperature strains is computed by comparing the member outside face temperature to its inside face temperature. Although the greatest temperature differential should theoretically be in the winter for a heated building, observations of precast insulated panels show that these panels usually bow outward, the effect of which would be increased by a higher **outside** temperature. A value of 100 deg. F for the outside face and an inside value of 75 deg. F is reasonable for most of the USA and Canada. Panels on South and West elevations tend to bow more than those on other sides, due to heat from the afternoon sun. For single-wythe panels, thermal strain is calculated across the full thickness of the member. For multi-wythe panels, the temperature differential is calculated from the centroid of each wythe. For example, with a 3-2-3 in. panel (8" thick), the differential will vary uniformly over 5 in., providing greater bow than an equivalent single-wythe (solid) section, which would be calculated using 8 in. If 0% composite is specified, then there are assumed to be no thermal restraint forces.

Multi-wythe panels can theoretically contain internal stresses due to differential temperature strain. Click on **Display internal wythe stresses…** to generate a graph of the stresses, assuming full composite action. These stresses may be reduced significantly when connectors are ductile, that is, they can creep over the course of the day. To account for this reduction, a "thermal stress connector creep reduction factor" can be applied. This value can range from 0 to 100%. 0% means the connector does not creep at all and there is no stress reduction. 100% means that the connector creep eliminates all the internal stresses due to differential temperature strain. Glass fiber connectors will generally creep more than carbon fiber connectors.

Capacity Check

<Back>

Ultimate and service load cases are specified and analyzed here. P-Delta effects, stress and deflection checks are made. Ultimate capacity is analyzed in the next screen (Interaction Curves).



Cracking Stress Coefficient:

The cracking stress coefficient (also known as the modulus of rupture) is the stress that is likely to cause cracking of the member. For normal weight concrete, ACI 318 recommends that the modulus of rupture be taken as 7.5 times the square root of F'c. (0.625 times the square root of F'c for Metric units.) Therefore, for normal weight concrete the appropriate coefficient to use would be **7.5**. See section 19.2.3.1 of ACI 318 for more information.

Design Load Check:

Click on a *Load Case* (1-14) to show the moment or stress diagrams due to suction and pressure. Click on the diagram at any point to list *Pu, Mu* and stresses at that point.

Press the **Show Stresses** button to display stresses at 100th points. Any areas where the tension stress exceeds the modulus of rupture (see above) are shown in red. If any areas are red then the section is cracked and the section properties have been reduced accordingly (see Methodology). This does not mean the member has failed, just that there will be increased P-Delta bow due to the use of a reduced moment of inertia in those areas. Compression stresses are to the left of the baseline (magenta color) and tension stresses are to the right (green color). The stress shown is the initial stress. Nearby cracking and/or P-Delta iterations could increase this tension stress beyond the crack stress, showing in red even though the initial stress may be below the cracking stress.

"Service Tension Exceeded" Warning:

This warning note appears (in red text) when service load tension exceeds 2/3 Mcr at the point selected. According to ACI 318 - R11.8.4.1, out-of-plane deflections increase rapidly when the service-level moment exceeds 2/3 Mcr (4/5 Mcr for prestressed members). LECWall now calculates this potential extra deflection automatically. If this warning occurs, reduce the Cracking Stress Coefficient to 5 (or 6 for prestressed members, English units) when checking service load cases, to induce cracked deflection. (These values are 0.42 and 0.5 respectively for Metric units.) Alternately, when using the Beam-Spring method, check the "Member is pre-cracked" check box. Compare the increased deflection to L/150. Maximum deflection (ignoring initial bow) due to service loads is limited to L/150 per ACI 318 - 11.8.1.1e.

Pre-Cracked Members:

Beam-Spring only: Mild-reinforced load-bearing panels are often analyzed as pre-cracked, so as to maximize the potential P-delta moments due to increased mid-point deflection. To check this when not using Beam-Spring, reduce the cracking stress coefficient (from the default 7.5) until the member shows as cracked.

Show Stress at Insulation Face:

The Beam-Spring method allows for the computation of interior wythe stresses at the insulation face of partially composite panels. These stresses are usually of lesser magnitude than at the exposed faces, but not always. If the Main Structural Wythe (MSW) is much thicker than the other wythe, then the MSW could attract enough moment to have greater tension stress at the insulation face then at the exposed face of the thinner wythe. If the panel shows as cracked but no red is visible on the graph, it could be due to insulation face cracking.

Second-Order Effects Ratio:

According to ACI 318-19 6.2.5.3 (6.2.6 for ACI 318-14), ultimate moments that include second-order (P-Delta) effects should not be greater than 1.4 times the ultimate moments due to first-order effects. This provision applies to slender columns only, and only for the critical load case. LECWall checks these moments at member mid-height.

Percent Composite:

When input manually, the program makes no assumptions about what percent composite action, if any, can be counted on for a particular member type. It is up to the engineer/user to provide reasonable values based on calculation, load tests and experience, noting that insulation bond can break down over time with a resultant loss of composite action. For this reason, percent composite for in-place loads should not depend on insulation bond. Panel designs that are to be tested to determine percent composite for in-place loads should have the insulation debonded between the panel wythes. (Insulation bond can be counted for handling stresses, however.)

The percentage of composite action for **Ultimate strength** is determined by the wythe connector's ultimate slip capacity. This percentage can be provided by the connector system manufacturer or calculated using LECWall's built-in Partial Composite Calculator. 100% composite action is recommended for Ultimate Strength. Otherwise the wythe reinforcing is not fully utilized. To achieve, consider increasing the number of connectors.

The percentage of composite action for **Stresses** is determined by the elastic *strain* in the wythe connectors. It is found by comparing the predicted partial composite cracking moment to the non-composite cracking moment and the fully-composite cracking moment. The partial composite section modulus is then taken as Mcr/fr, where Mcr is the partial composite cracking moment and fr is the ACI cracking stress based on 7.5*Sqrt(F'c). This percentage is typically less than the Ultimate Strength percentage at mid-span.

The percentage of composite action for **Deflection** is determined by the elastic *slip* in the wythe connectors, combined with the individual wythe stiffnesses. It is typically less than the Stress percentage.

See Partial Composite for the calculation method and assumptions used to determine the percentage of composite action. To calculate percent composite for a specific connector type and layout, use the built-in Partial Composite Calculator.

Click on the Interaction Curves button from the Capacity screen to display moment capacity vs axial load curves at a user-selected point. Click **Print Input** to print a three page report listing the problem input and showing section and plan views.

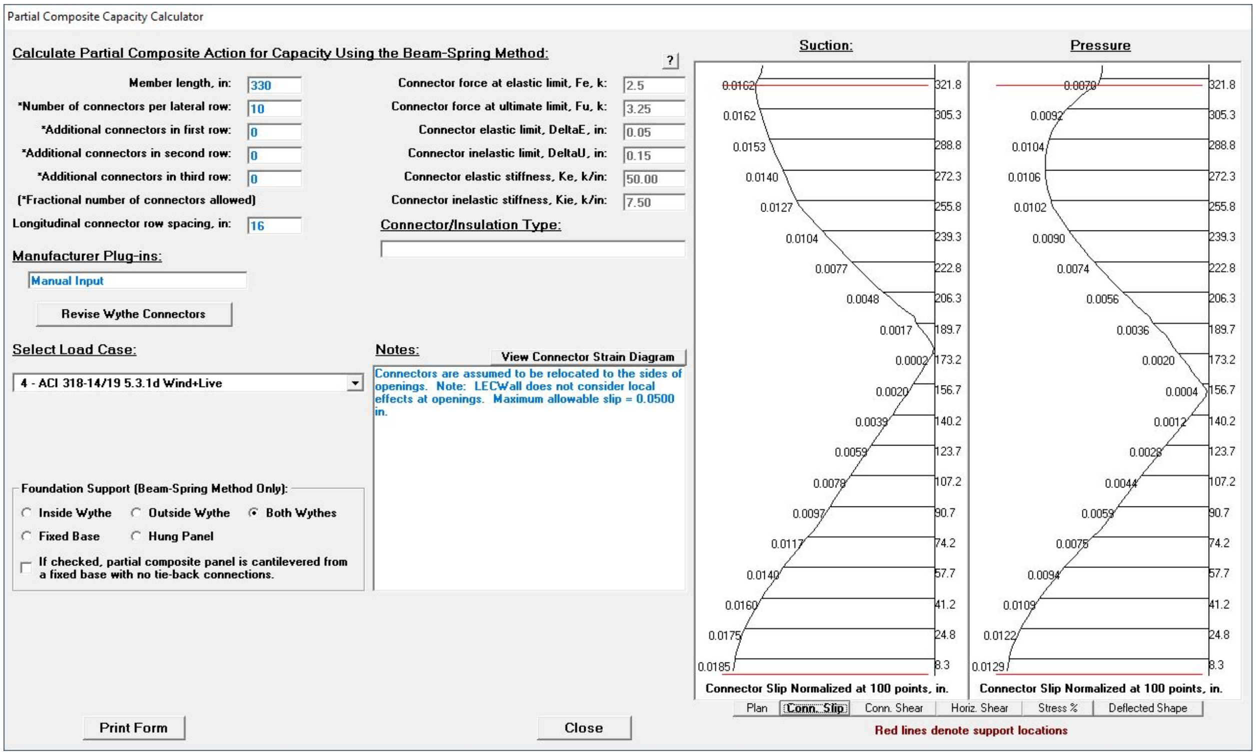
Click **Print Diagrams** to print moment and stress graphs at 100th points for a particular load case (1-14).

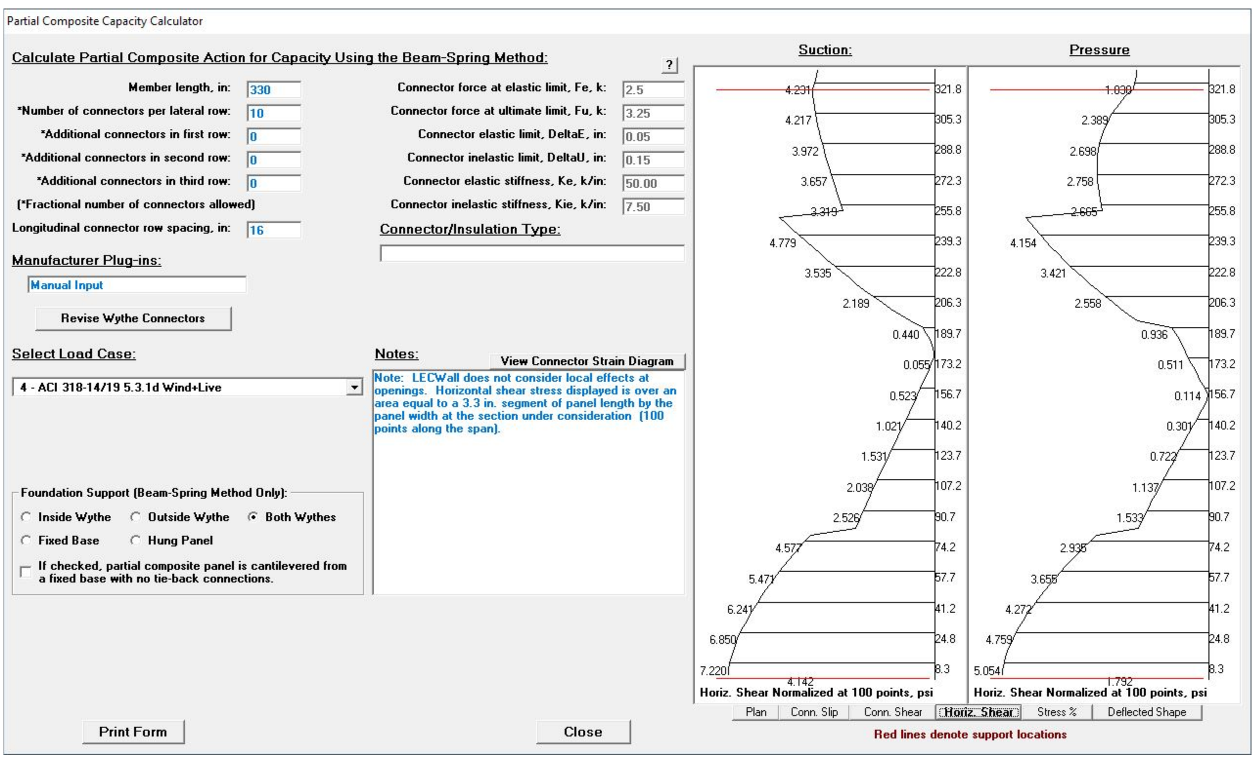
Beam-Spring Partial Composite Calculator:

<Back>

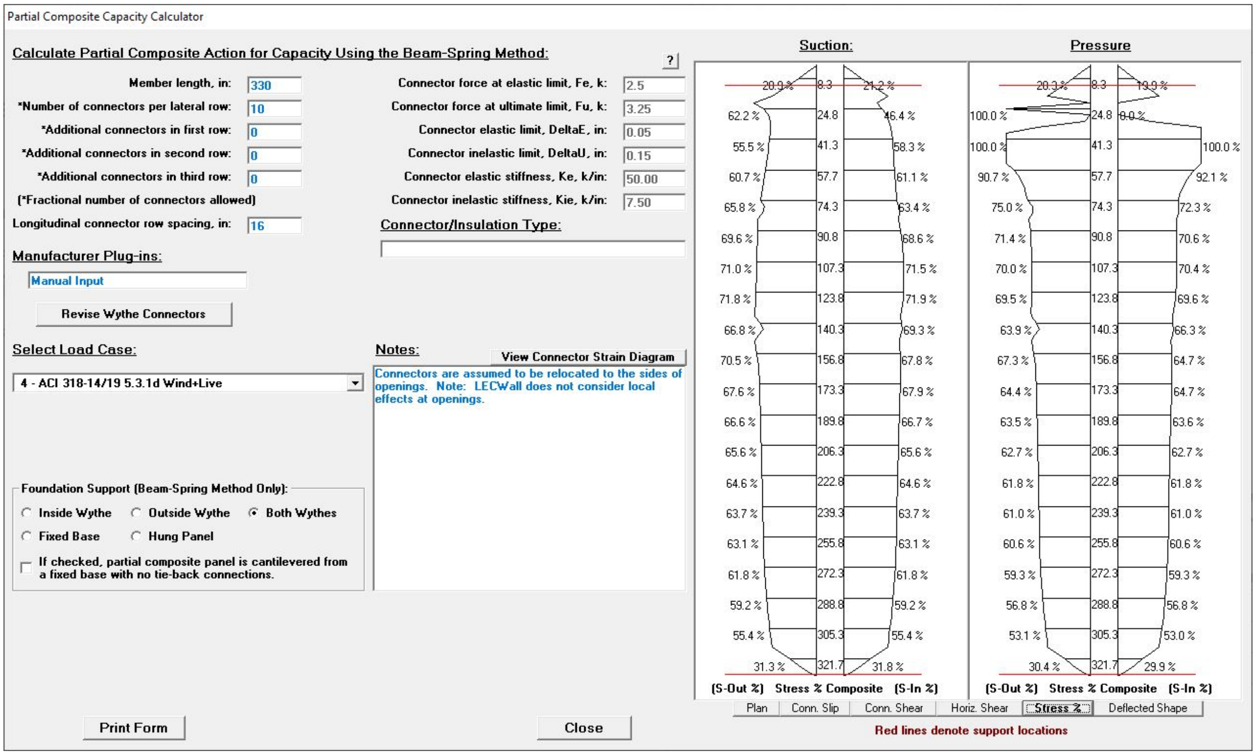
The ability of the wythe connectors to provide composite action varies according to their stiffness, location, type of load and support locations. Connector stiffnesses are divided over 100 points along the span for analysis. The Connector Slip graph shows the in-plane deformation between the wythes for the selected Load Case. Slip values show in red if the slip exceeds allowable limits. The slip limit for service loads is equal to Delta-E, the elastic limit. The limit for ultimate loads is the point where the connector shear force equals 0.75 Fu, per the PCI 150 Specification. This limit may be less for some connector types.

The shear in the wythe connectors is directly proportional to the connector slip. The shear force displayed is not per connector, but is for an area equal to 1/100 of the panel length by the full panel width. Likewise, the horizontal shear stress graph shows the average horizontal shear stress for the same area. This is useful for determining the horizontal shear stress in the solid zones, which must average below 60 psi (410 kPa) to be considered.

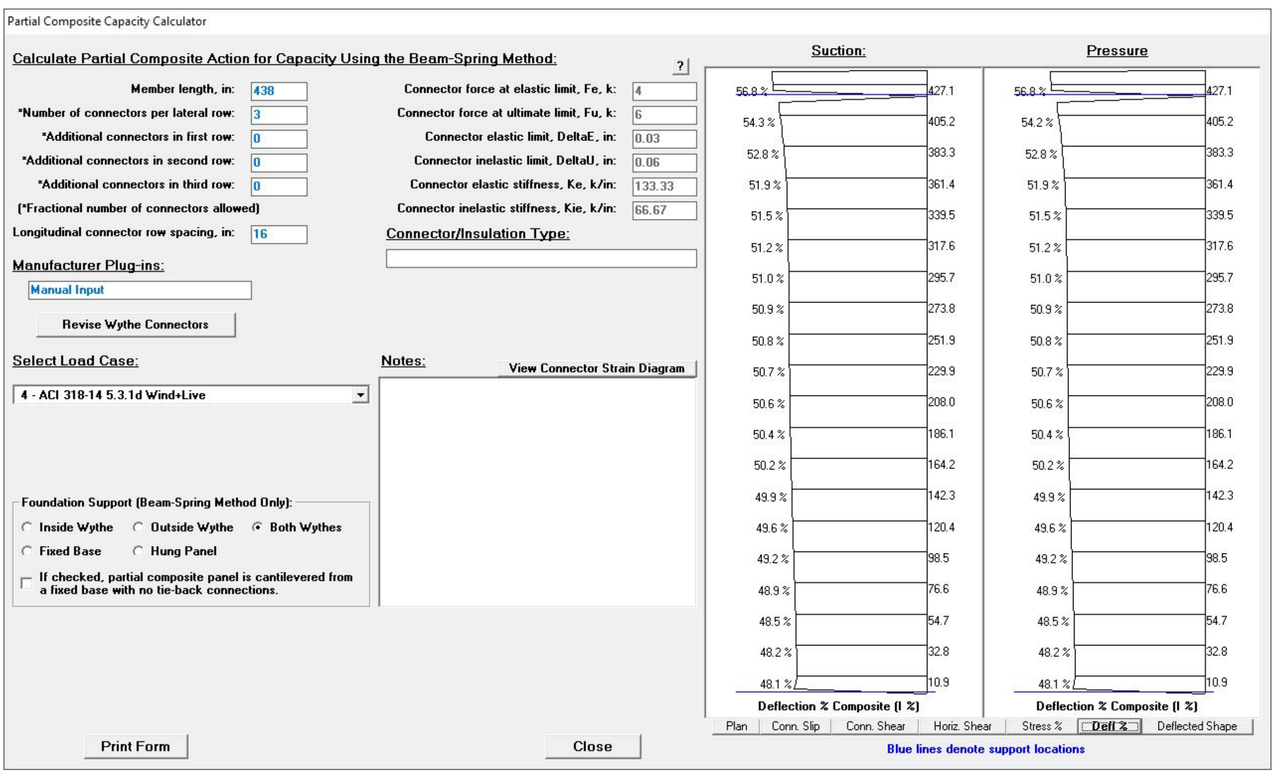




The Stress % Composite graph is used to find 1.0Mcr. This value varies across the span depending on wythe connector stiffness, loading condition and support locations.



The Deflection % Composite graph is provided for information only. As with Stress % Composite, this value varies across the span depending on wythe connector stiffness, loading condition and support locations.



ISBT Partial Composite Calculator
(Academic and Manufacturer Versions Only)

<Back>

ISBT stands for "Iterative Sandwich Beam Theory" and is based on the following paper:

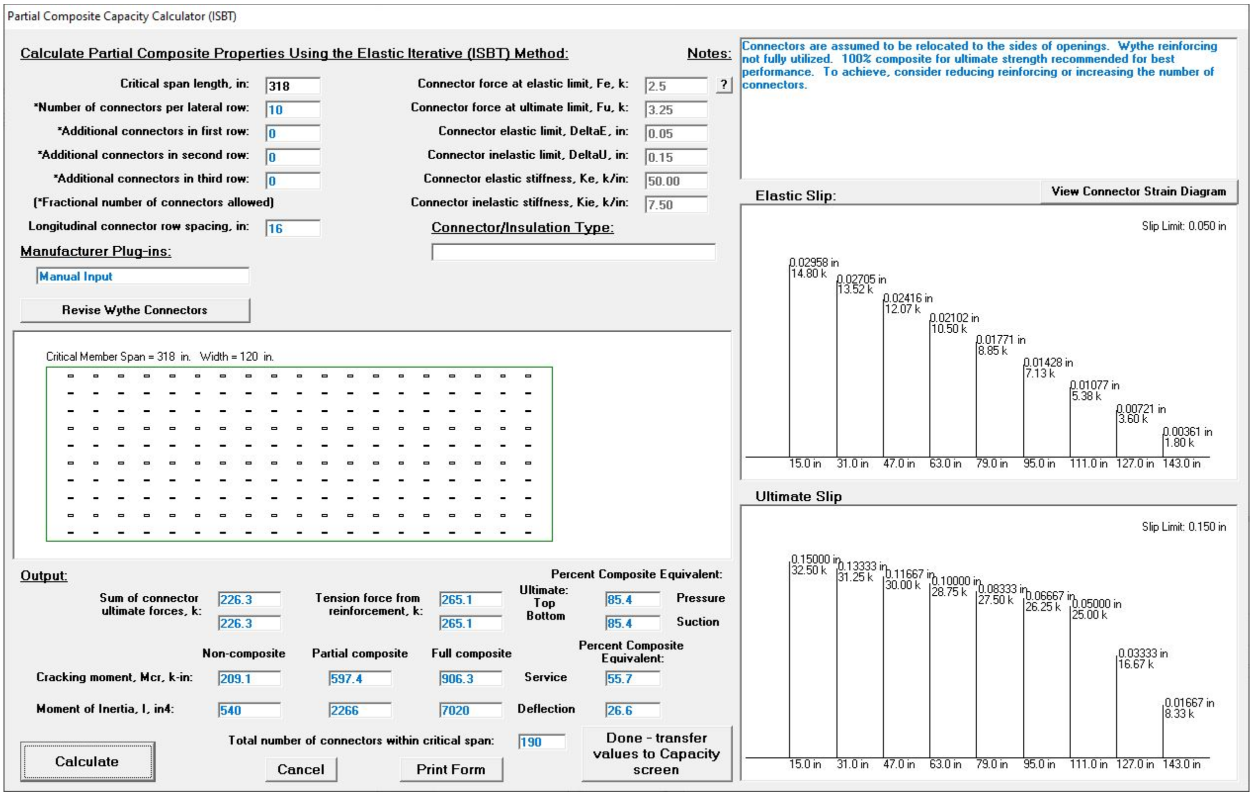
"Iterative and Simplified Sandwich Beam Theory for Partially Composite Concrete Sandwich Wall Panels", Journal of Structural Engineering, Volume 147 Issue 10 - October 2021, Salam Al-Rubaye, S.M.ASCE; Taylor Sorensen, A.M.ASCE; and Marc Maguire, A.M.ASCE.

The Beam-Spring method is more flexible and accurate than the ISBT method, especially when the condition is anything other than a uniform load over a simple span. Therefore, for most situations, Beam-Spring is preferred over ISBT method. ISBT does have its uses though. It is amenable to hand calculation, and so provides an independent verification or double-check of other methods. It can also be used to analyze custom connectors. For example, by measuring deflection and stress levels in a test panel and converting to equivalent % composite values, the connector elastic stiffness (Ke) can be found through trial and error.

The four connector property values (or characteristics) are provided by the connector manufacturer, and are based on a standard double-shear testing method as specified by the Precast/Prestressed Concrete Institute. The test method is expected to become part of a new ANSI standard for partially-composite insulated walls. Click on "Revise Wythe Connectors" to enter or change these values.

The critical span length is usually the span where the maximum moment occurs. It is not the same as the panel length. It is the center-to-center distance between lateral supports. LECWall makes an educated guess as to the span, but the user may need to over-ride if there are significant non-uniform loads that make another span critical. Note that it is conservative to use a shorter span length, as this will reduce the degree of composite action (there are fewer connectors to engage).

A rational elastic iterative process is used. Uniform load is assumed for this "pre-analysis". Additional connectors may be required near the base (beyond what is shown by LECWall) if earth pressure load is the critical case (or use Beam-Spring instead). If the critical span is a cantilever, then all extra end connectors (from both ends) should be located at the cantilever base instead, doubling them up (or use Beam-Spring, which has a cantilever option). Both the ISBT and Beam-Spring methods find the percentage of tension reinforcement that can be transferred between wythes by the wythe connectors, using a linear strain distribution at each point along the panel. This then becomes the % composite for Ultimate value.



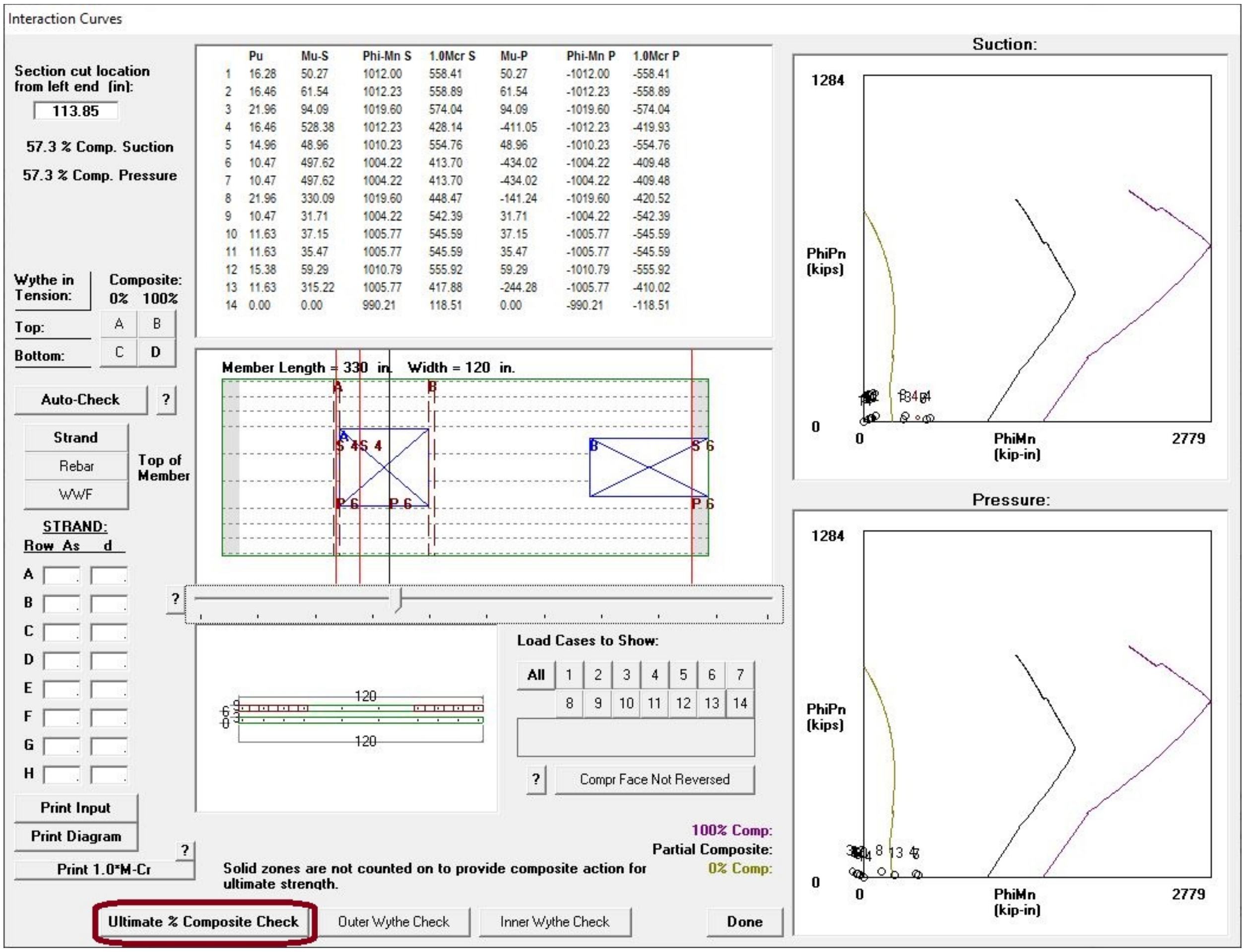
Click on the Interaction Curves button from the Capacity screen to display moment capacity vs axial load curves at a user-selected point. Click **Print Input** to print a three page report listing the problem input and showing section and plan views.

Click **Print Diagrams** to print moment and stress graphs at 100th points for a particular load case (1-14).

Interaction Curves

<Back>

Click on the Interaction Curves button from the Capacity screen to display moment capacity vs axial load curves at a user-selected point.



Slide Bar:

Move the slider below the member plan view to adjust the section cut location. Use the keyboard left or right arrow to move the bar at 1/100th increments of the member length. Click just to the left or right of the slide button to move in larger increments. You can also move the mouse pointer onto the button and press and hold the left mouse button to "grab" the button, then slide the button to the exact point desired.

As the bar is moved, the interaction curve below is updated automatically to allow a quick search for critical locations. Alternately, click the **Auto-Check** button to find critical locations automatically. The load case no. and "P" for pressure, or "S" for suction will be displayed on the plan view.

The compression block is shown as 25 rectangular red blocks in the section view. Use this view to verify that blockouts and reveals are input correctly and that the program has taken them into account in sizing the compression block. Click on A thru D to view various combinations of 0% and 100% composite designs under suction or pressure.

Click the **Strand** button to check total developed strand area and "d" distance to the top of the compression block. Likewise, click the **Rebar** or **WWF** buttons to check the mild steel area and location at the section cut and compression block orientation selected.

Load Cases to Show:

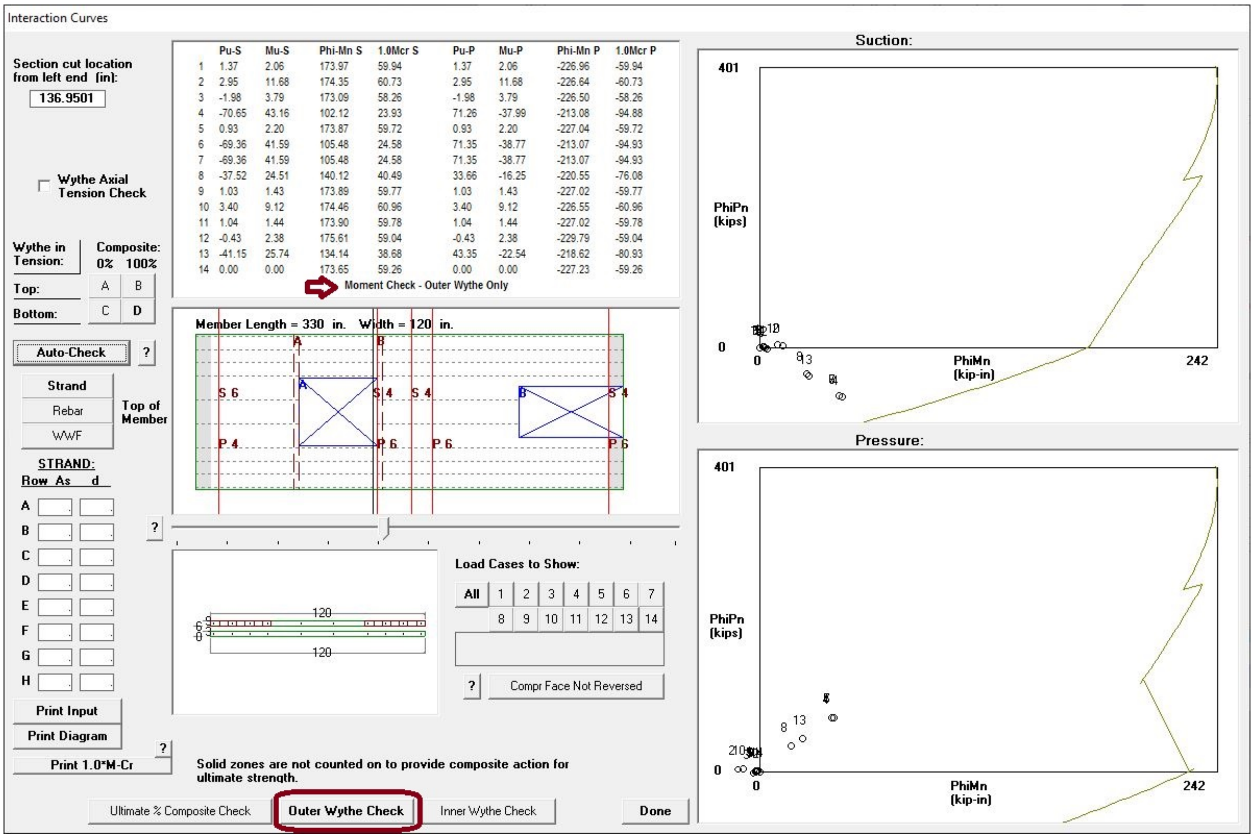
Select **All** to superimpose all fourteen load cases on the interaction curve, or select each individually by clicking the button with the load case no. desired. The compression face defaults to the face which is usually in compression (assuming single curvature) from the applied wind load moment (either suction or pressure). If gravity loads with a high negative eccentricity are used or the optional floor connection is specified, the compression face assumed by the program may not be the critical face for design. If negative moments, or no moments, are displayed for some or all load cases it is likely that this is the case. Click on the **Compression Face Not Reversed** button to reverse the compression face and check the interaction curve again for critical load cases. Be sure to check wind pressure as well as suction by clicking the **Pressure** button.

1.0 * Cracking Moment:

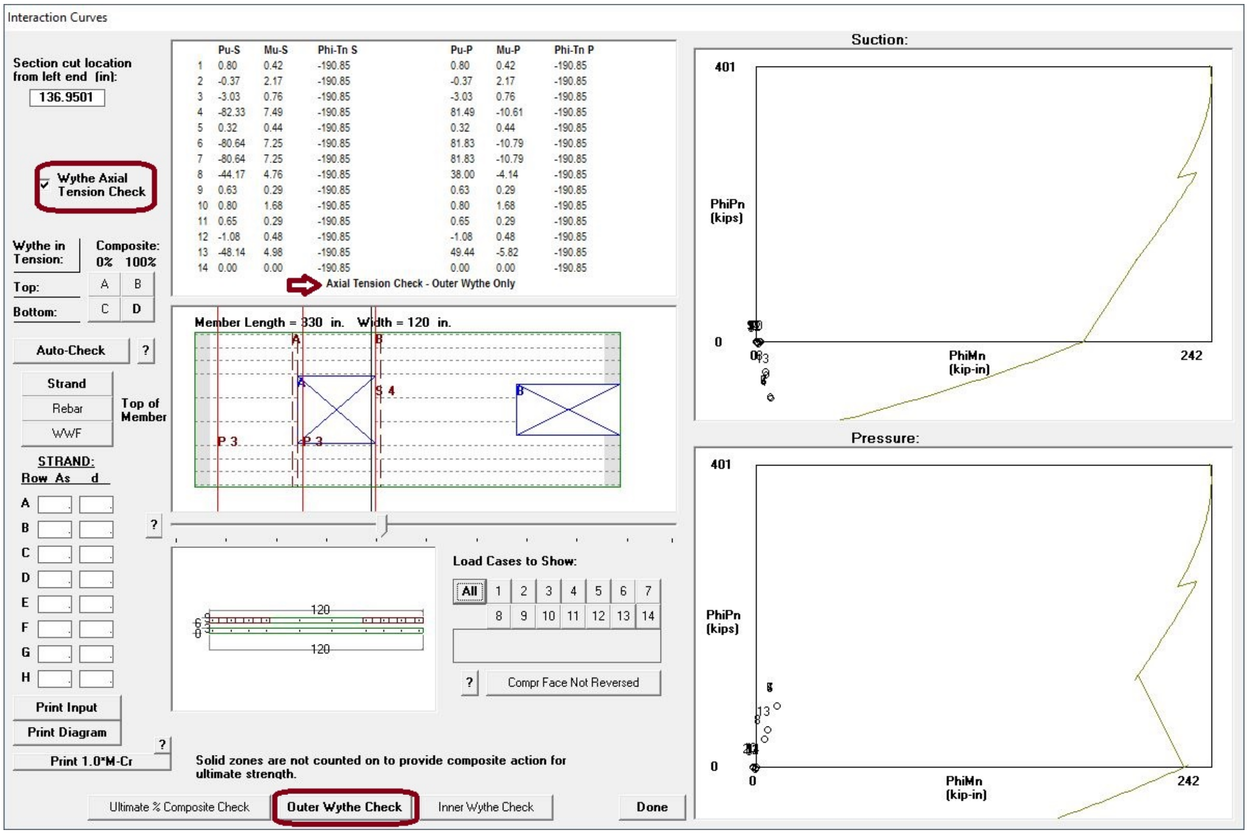
1.0 times the theoretical cracking moment is displayed by a small red dot for each load case. (See ACI 318, Section 11.8.1.1c) This can be suppressed on the printout if desired by selecting the '*Do not print 1.0 Mcr' button*." By keeping the member capacity greater than 1.0 MCr, additional capacity will be present after cracking, which should provide a "warning" deflection before failure occurs. According to common practice, if member flexural strength is at least twice the applied factored load, then the ultimate capacity can be less than 1.0 MCr. Note that this criteria only applies at the point where the member is likely to crack *first*, not along the entire length. For mild-reinforced sections 1.0 Mcr is also used. If the applied moment is less than half of the 1.0 cracking moment, the 1.0 Mcr value is not plotted.

Individual Wythe Checks:

Beam-Spring only - select **Outer Wythe Check** or **Inner Wythe Check** to view the behavior and capacity of each wythe individually. This is more accurate than the **Ultimate % Composite Check** when there is a cantilever or intermediate supports.



It's possible that wythe axial tension could govern over the wythe moment capacity. The Wythe Axial Tension Check option re-runs the analysis with the wythes assumed fully cracked (to maximize axial tension) and compares the axial tension in the selected wythe to the developed reinforcing capacity. Note that this is not a substitute for selecting the Pre-Cracked option in the Capacity screen. The Pre-Cracked option is still required to model secondary (P-Delta) effects for a slender wall analysis of mild-reinforced panels.



Sections:

See Sections to view a section cut at any point along the member with reinforcing and compression block.

Printing:

The Print Input button is provided as a convenience to allow printing of the input data from this window.

Click on **Print Diagrams** to print a one page display of the interaction curves for the selected point along the member. For multi-wythe members, the 0% composite curve is shown as a dotted line and the 100% composite curve is shown as a dashed line.

Sections

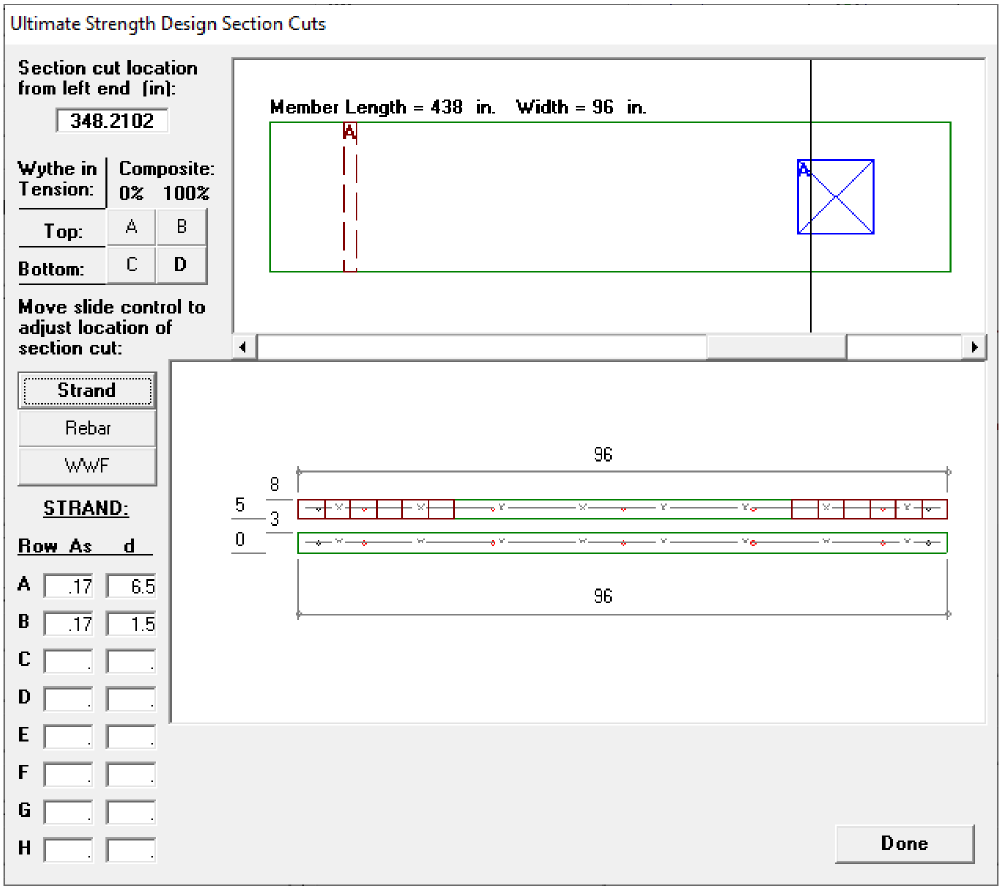
<Back>

Ultimate Strength Design Section Cuts:

Click **Sections** from the Capacity Check window (or Interaction Curves screen) to view a section cut at any point along the member. Use the slide bar below the member plan view to select the section cut to view.

The compression block is shown as 25 rectangular red blocks. Use this view to verify that blockouts and reveals are input correctly and that the program has taken them into account in sizing the compression block. Click on A thru D to view various combinations of 0% and 100% composite designs under suction or pressure.

Click the **Strand** button to check total developed strand area and "d" distance to top of compression block. Likewise, click the **Rebar** or **WWF** buttons to check the mild steel area and location at the section cut and compression block orientation selected.



Units

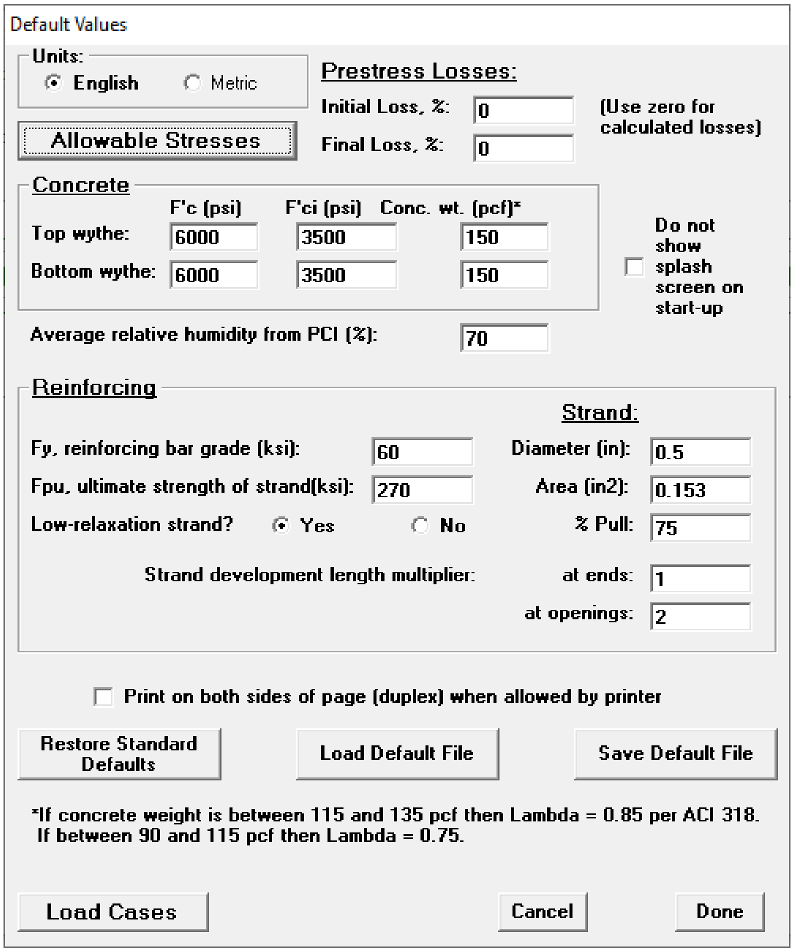
<[Back](#)>

Allows selection of English or Metric units for a particular problem. The default units for all *new* problems can be specified in the *Setup* pull-down menu under [Defaults](#).

Defaults

<[Back](#)>

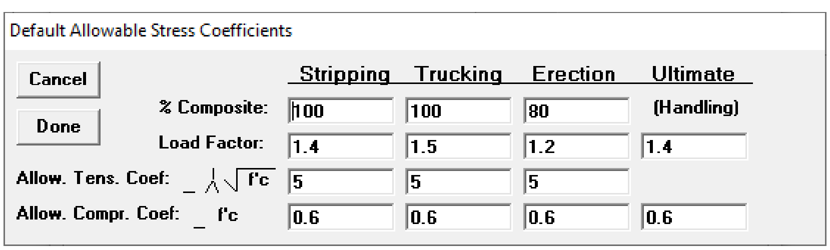
Default Values:



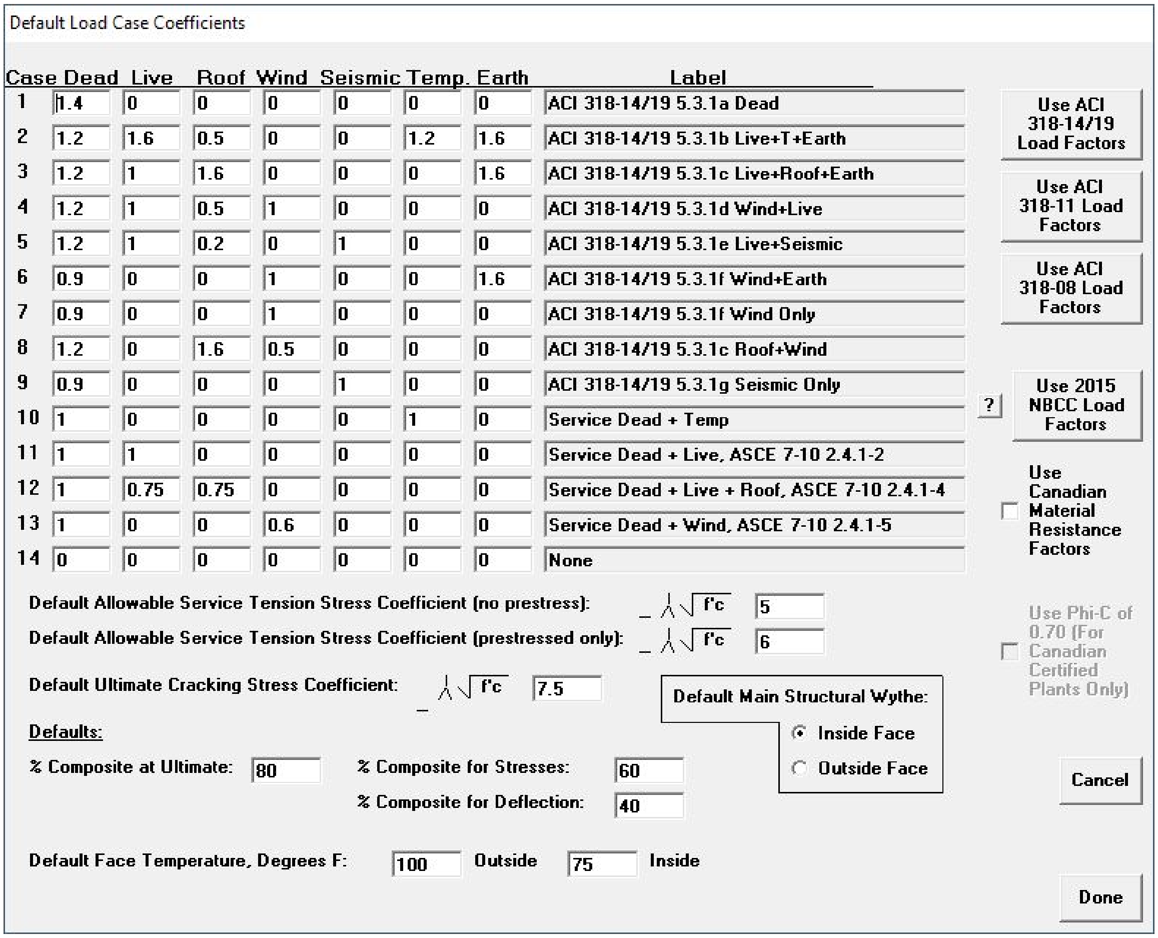
Most of the constants used in this program can be customized by the user. Click on **Setup** from the pull-down menu and select **Defaults**.

When a new file is started, many of the inputs are predefined from this window. The default values are stored in the file CWDEFALT.DAT in the LECWALL directory. If this file is deleted or moved, it will be recreated by the program using the values provided when first installed (the "factory" settings).

Here, Units can be set to default to English or Metric for all *new* problems. Concrete and reinforcing properties can also be predetermined.



The above coefficients and factors apply to the Handling analysis only. See below for the default values used for Capacity calculations:



Allowable stress coefficients and load cases can also be modified from the preset ACI values, if, for example, a different Code is used. The Service Tension Stress Coefficient (mild reinforced) defaults to 5 (0.417 times the square root of F'c for Metric units), per the PCI Design Handbook Equation 5-7, assuming "no discernable cracking" as the criterion. The Service Tension Stress Coefficient for prestressed defaults to 6 (0.5 times the square root of F'c for Metric units). While ACI 318 Table R24.5.2.1 allows 7.5 for Class U prestressed members, it is common in the industry to use a lesser value to provide a factor of safety against face cracking. If *Use Canadian material resistance factors* is selected, the Canadian Code, CSA A23.3 will be used for ultimate capacity determination.

Troubleshooting

<Back>

- **"Error reading file LECWALL.LF" message**
  The license file, LECWALL.LF, was not found and the program will not run. If the file has been lost or deleted, re-install LECWall by running the downloaded setup program. Verify that the splash screen shows the correct version and latest revision date.

Frequently Asked Questions

<Back>

Here are some common customer questions and answers:

What is “Tilt Potential,” as shown on the Handling screen?

LECWall estimates the degree of tilt in the panel if all the lines are on rollers and all the rollers are released. It is based on the center of gravity of the panel in relation to the center of all the lift points. This is useful for erection planning. The center of gravity of the panel is listed near the top of the Handling window. Use the C.G. to assist in locating the lift points to provide a more equal distribution of lift forces. Top of panel is shown on the left, so lifters could be positioned to induce a clockwise tilt, which would bring the top of panel up as the panel is lifted from the trailer.

While running LECWall, I came across two questions about how the program considers prestressing forces:

1. While doing a P-delta analysis, which load cases consider the axial force due to prestressing?

2. What load factor is used for prestressing force?

1. Axial force due to prestress is present for all load cases. It does not contribute to P-Delta effects directly. If prestress is eccentric and induces bow, then Delta will increase, causing a net greater P-Delta due to other applied gravity loads. Even so, the prestress force always follows the curve of the panel and is internal to the panel, so does not contribute any P-Delta force of its own.

2. No load factor is used for the prestress force (Load Factor = 1).

Reinforcing note: The reinforcing input unit is bar **diameter**, not area. If bar area is input by mistake, that could show the member as being under-reinforced.

When running a service load case, sometimes stresses show as red on the diagram even though the section is uncracked at that location. Why is that?

For service load cases, LECWall uses the “Allowable Service Tension Stress Coefficient” specified on the “Load Case Coefficients” screen (attached). This defaults to 5 sqrt F’c for mild reinforced members, or 6 sqrt F’c for prestressed. So, although the member is not cracked, tension stresses exceed that recommended by PCI for serviceability when exposed to view and weather (see PCI Handbook, 8th Ed. Example 5.2.2.1).

How does LECWall calculate the cracking moment for a wall panel? Are any load factors included?

Self-weight, applied axial load and prestress force all increase the cracking moment. Load factors are also applied, consistent with the load case being considered. Sometimes mild steel needs to be added to prestressed members to satisfy the ACI 1.0Mcr requirement. The requirement initially applies at the location most likely to crack first.

How should I specify lifter locations for the situation where there are double rollers for one of the lines without a spreader bar?

LECWall assumes two lines of support to maintain static determinacy. See Optional Sub-Gaps for the procedure to add additional rollers.

When using the Beam-Spring method, why does the Capacity screen sometimes call out the section as cracked while displayed stresses are still below the cracking stress?

The stress display could be confusing in that the stress shown is from the initial analysis before the effect of P-Delta deflections and redistributions due to a cracked section. The reason LECWall doesn’t show the final stress is that this is not always a real number – cracking the panel redistributes flexural stresses to the connectors and adds additional axial tension in the tension wythe. It is a dynamic, non-linear process. For this case, the crack stress was likely exceeded in an intermediate run, then dipped down again after cracking was taken into account, reducing the wythe stiffness.

LOSCH SOFTWARE LICENSE AGREEMENT

<Back>

This is a legal agreement between you (the Licensee) and Losch Software, Ltd (the Licensor).

1A. TRIAL LICENSE GRANT. Losch Software, Ltd (the Licensor) grants to you the right to use one copy of the Licensor software program(s) (the "Software") on a single computer for a period of 15 days from receipt of the Software, in exchange for the consideration of your evaluation of the Software for purchase. You may not network the Software. Trial version printouts are not to be disseminated or included in a calculation package. The printouts are to be used in-house only, for purpose of Software evaluation. By the end of the evaluation period, you agree to either A) license the Software under the terms of a separate license agreement provided by Licensor, or B) stop using the program and erase any and all copies.

1B. SINGLE LICENSE GRANT. The Licensor grants to you the right to use one copy of the Software on a single computer. You may not network the Software or otherwise use it on more than one computer or computer terminal at the same time without prior arrangement with Licensor.

1C. MULTIPLE LICENSE GRANT. The Licensor grants to you the right to install and use the Software on up to the licensed number of named seats or stations at the site address registered with the Licensor. Installing or accessing the Software off-site would be a violation of this agreement, except for portable computers used by Licensee's employees.

1D. COMPANY-WIDE LICENSE GRANT. The Licensor grants to you the right to install and use the Software on up to the licensed number of named seats or stations at multiple sites.

2. COPYRIGHT. The Software is owned by Licensor and is protected by United States copyright laws and international treaty provisions. Therefore, you must treat the Software like any other copyrighted material except that you may either (a) make one copy of the Software solely for backup purposes, or (b) transfer the Software to a single hard disk provided you keep the original solely for backup purposes.

3. OTHER RESTRICTIONS. You may not rent, lease or otherwise distribute the Software, but you may transfer the Software and accompanying written materials on a permanent basis provided you retain no copies and the recipient agrees to the terms of this Agreement. You may not reverse engineer, decompile, or disassemble the Software. THE SOFTWARE IS TO BE USED SOLELY BY A LICENSED PROFESSIONAL OR STRUCTURAL ENGINEER FAMILIAR WITH PRECAST/PRESTRESSED CONCRETE DESIGN.

4. LIMITED WARRANTY. Licensor offers no warranties with respect to the operation or fitness for a particular use.

5. NO OTHER WARRANTIES. LICENSOR DISCLAIMS ALL WARRANTIES, EITHER EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE, WITH RESPECT TO THE SOFTWARE AND THE ACCOMPANYING WRITTEN MATERIALS.

6. NO LIABILITY FOR CONSEQUENTIAL DAMAGES. IN NO EVENT SHALL LICENSOR BE LIABLE FOR ANY DAMAGES WHATSOEVER (INCLUDING, WITHOUT LIMITATION, DAMAGES FOR LOSS OF BUSINESS PROFITS, BUSINESS INTERRUPTION, LOSS OF BUSINESS INFORMATION, OR OTHER FINANCIAL LOSS) ARISING OUT OF THE USE OF OR INABILITY TO USE THIS SOFTWARE, EVEN IF LICENSOR HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

This Agreement shall be governed and interpreted by the laws of the State of Nevada.

(2024-10-23)