

LECWAll - Precast/Prestressed Sandwich Wall Design

Contents

	<u>Introduction</u>
	<u>Methodology</u>
File:	<u>New File</u> <u>Open File</u> <u>Save File</u> <u>Save File As...</u> <u>Delete File</u> <u>Print Input</u> <u>Exit</u>
Section:	<u>Section Dimensions</u> <u>Materials</u> <u>Reinforcing</u> <u>Openings</u> <u>Reveals</u> <u>Additional Weight</u>
Losses:	<u>Prestress Losses</u>
Handling:	<u>Handling Check</u> <u>Change Allowable Stress</u> <u>% Composite</u>
Capacity:	<u>Capacity Check</u> <u>Interaction Curves</u> <u>Sections</u>
Units:	<u>Units</u>
Setup:	<u>Defaults</u> <u>Troubleshooting</u> <u>Frequently Asked Questions</u>
Misc:	<u>License Agreement</u>

Introduction

This program is designed to assist the licensed engineer in the design of precast and/or prestressed 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, user specified from 0% to 100%.
- Up to 8 openings or notches of any size accomodated 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 or CSA A23.3.
- 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.
- 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. To print it out, Click on Methodology in this paragraph, select **File** from the Help screen menu, then select **Print Topic**.

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 main window can be moved around by pointing to the title bar while holding down the left mouse button. Drag the window to the desired location and release the mouse button. 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.
2. Select Section Dimensions from the **Section** 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 **Section** 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 **Section** menu. There are three categories of reinforcing to fill in: strand, rebar and/or welded wire fabric.
5. Select Openings from the **Section** menu if there are any openings or notches in the member.
6. Select Reveals from the **Section** menu if there are any reveals or depressions on the front or back face of the member.
7. Select Additional Weight from the **Section** menu if there are any significant areas where insulation is cut out or any concrete haunches are present.
8. To examine or change calculated prestress losses, select Prestress Losses from the **Losses** menu.
9. The input file can be saved at any time by selecting Save File from the **File** menu.
10. Select Handling Check from the **Handling** menu to check stresses and ultimate capacity at stripping, during transportation or at erection. Select **Print Formatted** to print input and handling graphs (two pages of input plus three pages of graphs).
11. 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. Select each load case (1 thru 6) then click on **Show Stresses** to check for a cracked section. Click on **Print Diagrams** to print the moment and stress charts, if desired. Select **Interaction Curves** to check capacity at critical points for all six load cases. The **Autocheck** button can be used to find critical locations automatically. For each critical case found, 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.

Methodology

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 [% 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. We recommend a value of 1.0 at ends and 2 (or higher) at openings, to account for the differing strains near the opening.

Prestress Losses:

The PCI 7th Edition Handbook is used for calculation of initial and final [Prestress Losses](#) in the member (Section 5.7).

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.

1.2 times the theoretical cracking moment can also be displayed, per ACI 318, section 18.8.2. By keeping the member capacity greater than $1.2M_{Cr}$, additional capacity will be present after cracking, which should provide a "warning" deflection before failure occurs. According to ACI, if member flexural strength is at least twice the applied factored load, then the ultimate capacity can be less than $1.2M_{Cr}$. Note that this criteria only applies at the point where the member is likely to crack **first**, not along the entire length. Also, the ACI code provision is meant for in-place loadings, not temporary handling forces. Therefore, it is up to the engineer's discretion as to whether this criteria needs to be applied for handling loads.

P-Delta Analysis:

The program does an iterative second-order analysis of the member under each of the six specified load cases. The PCI 7th Edition Handbook procedure (Section 5.9.3.1) 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 conservatively calculated using an approximate procedure developed by Antoine Naaman for partially prestressed members ("Prestressed Concrete Analysis and Design Fundamentals", McGraw-Hill, 1982; also page 98 of the May-June 1989 issue of the PCI Journal). 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 the slope-deflection method. 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 is considered to be a first floor, at grade connection. Wind loads are applied above the floor connection only (a conservative assumption), therefore it would not be valid to use the floor connection for an upper floor (above grade). 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.

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.

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 7th Ed. Handbook, Fig. 5.9.1. A straight line is plotted from the transition point of $\Phi P_n = 0.10f'_c A_g$, where $\Phi = 0.7$ (or 0.65 for ACI 318-08 or 318-11), to $\Phi P_n = 0$, where $\Phi = 0.9$. 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.

Partially composite capacity is found by first computing the capacity at 0% composite and again at 100% composite, then finding a ratio based on the percent composite specified (see % Composite). The 0% and 100% capacity curves are plotted along with the partially composite curve for comparison purposes.

New File

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.

F'_c

F'_c = Concrete ultimate compressive strength at 28 days

F'_{ci} = Concrete ultimate compressive strength at release

Delete File

To permanently delete a selected member file from the hard drive: Select Open File or Save File As..., click on the file to highlight it, and press the Delete key on the keyboard.

Exit

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

Materials

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 will be "ghosted" and will not allow any input. For this reason, the Section Dimensions form should be filled out before the **Materials** form.

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 **Done** to close the form or **Cancel** to start over.

Section Dimensions

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*.

The bottom wythe is required and cannot be zeroed out. Stems and hollow-core openings can only be placed on the bottom wythe. For this reason, the bottom wythe should usually be selected as the main structural wythe, unless the top wythe is stiffer. The program will warn you if the selected wythe is not as stiff as the other wythe. This is important when modelling partially 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) 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 percent of composite action specified by the user.

The section drawing on the main form is updated when you click **Done**. Look at this drawing to verify that the dimensions were input correctly.

Reinforcing

This form contains all the information related to the strand, rebar and welded wire fabric in the section. Up to four rows of each can be specified (A,B,C or D). Click on button **A** to input Row A, etc. If more than four 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).

Reinforcing bars do not need to extend the full length of the member. Input a start point from bottom and from top, as desired.

For columns only: If column ties are not provided, the program will multiply calculated capacities by a factor of .85, in accordance with the ACI Recommended Practice. (See the PCI Design Handbook, 4th Edition, pg 4-46.) A column can be defined as a load-bearing member with a reinforcing ratio greater than .01, and an average prestress stress of less than 225 psi. If the member is a wall panel, and is not defined as a column by the above criteria, it is acceptable to check the Yes box as if column ties are provided, even if they are not.

Print Input

This command will send a two 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.

Openings

This form displays all openings and notches in plan view in blue. (reveals are displayed in red for reference.) Up to eight openings can be specified (A-H). **Openings should not overlap with other openings or reveals.** Click with the mouse 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 section view to assist in locating bars and strand to miss openings (see Reinforcing - Locate Horizontally). Use the **Update** button to view any changes made to openings.

Reveals

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 eight reveals can be specified (A-H). **Reveals should not overlap with other reveals or openings.** Reveals can run longitudinally or transverse. Click with the mouse 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

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 eight loads can be specified (A-H). 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

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

The Handling Analysis screen has three parts. The first is a shear and moment display. It is called up by clicking the **Shear/Mom** button after selecting the **Stripping, 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.

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. **A note of warning:** *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.

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. **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). Bars above the baseline denote a net tension stress (positive). Red 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.

Check of Ultimate Capacity for Handling Forces:

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](#). 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) for a proper design.

1.2 times the theoretical cracking moment can also be displayed. 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.

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 the 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 the **Print Handling Stress Graphs** check box to print the stress graphs for the selected conditions. Click on the **Print Handling Ultimate Capacity Graphs** check box to print ultimate capacity graphs for each selected condition.

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 λ and F_c or F'_{ci} . The tension coefficient is multiplied by λ and the square root of F_c or F'_{ci} . The default values are as recommended by the PCI Design Handbook, Section 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.

% Composite

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 ("MI") would be:

$$MI(0\%) + (MI(100\%) - MI(0\%)) * .80$$

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.

Some references regarding sandwich panel composite action are:

- 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,

Capacity Check

Applied Loads:

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. If a mezzanine is present, a different wind pressure can be specified for the panel area below the mezzanine. If the intermediate connection is for a slab-on-grade, then no wind load is applied below the connection.

Superimposed Dead Load is dead load that does not include the self-weight of the member. 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. The same applies for **Applied Live Load**. If multiple loads with

different eccentricities are present, use an average e for the total. Load 1 is applied at the roof connection. Load 2 is applied at the optional mezzanine connection.

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. For Release 11 only (.WAL files), input negative values for pressure from outside, such as at a basement area below grade. For Release 12 (.W12), both outside and inside pressures should always be positive, acting toward the wall.

For single story files only (.WAL), if the optional slab-on-grade floor connection is used, earth pressure is automatically assumed to be applied from the floor connection to the base of the panel. Therefore, the distance to top of retained earth should be input as zero to avoid an error message. If the floor connection is for a mezzanine, or the only connection is at the base of the panel, input the top of slab or grade level as requested.

For multi-story files (.W12), input the top of grade level from base of member as requested.

Floor tie not active when lateral 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.

Support Locations:

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.

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. If the floor connection is specified as for a mezzanine, wind load is applied below the connection and superimposed gravity loads are allowed at the connection height. **Slenderness effects are not calculated when a mezzanine is used.** (The user should check that the clear span of the member (h) does not exceed the recommended ACI h/t ratio.)

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.

Initial Bow & Temperature Strain:

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 sandwich panels show that these panels always bow outward, the effect of which would be increased by a higher **outside** temperature. A value of 100 to 110 deg. F for the outside face and an inside value of 70 to 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. Thermal strain is always calculated across the full thickness of the member, regardless of the percent of composite action. If 0% composite is specified, then thermal restraint forces will be calculated based on the non-composite properties. If greater than 0% (even .01%) composite is specified, then 100% composite properties will be used for calculation.

Percent Composite at Ultimate:

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 ultimate should not depend on insulation bond. Panel designs that are to be tested to determine percent composite for ultimate should have the insulation debonded between the panel wythes.

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 9.5.2.3 of ACI 318 for more information.

Slenderness Effects:

Check the **Yes** box to enable P-Delta analysis. Check the **No** box to disable P-Delta analysis. P-Delta analysis can be disabled if the member is laterally restrained against bow more or less continuously along its height. For example, a corner wall panel with multiple corner connections could be considered as laterally restrained. A member with a mezzanine connection is automatically considered to be restrained. Otherwise, slenderness effects should always be included.

Design Load Check:

Click on a *Load Case* (1-12) to show the magnified moment diagrams due to suction and pressure. Click on the diagram at any point to list P_u , M_u 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](#)). Compression stresses are to the left of the baseline and tension stresses are to the right.

Interaction Curves:

Click on the [Interaction Curves](#) button to display moment capacity vs axial load curves at a user-selected point. Click **Print Input** to print a two page report listing the problem input and showing section and plan views.

Click **Print Diagrams** to print magnified moment and stress graphs at 100th points for a particular load case (1-12).

Ultimate Strength Design Section Cuts:

Click [Sections](#) to view a section cut at any point along the member. Use this for verifying input or noting compression block locations.

Slide Bar

Click on the 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.

Interaction Curves

Click on the **Interaction Curves** button from within the [Capacity Check](#) window to display moment capacity vs axial load curves at a user-selected point.

Move the [Slide Bar](#) below the member plan view to adjust the section cut location. 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 twelve 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.

1.2 * Cracking Moment:

1.2 times the theoretical cracking moment is displayed in red for each load case. (See ACI 318, Section 18.8.2.) By keeping the member capacity greater than $1.2M_{Cr}$, additional capacity will be present after cracking, which should provide a "warning" deflection before failure occurs. According to ACI, if member flexural strength is at least twice the applied factored load, then the ultimate capacity can be less than $1.2M_{Cr}$. Note that this criteria only applies at the point where the member is likely to crack *first*, not along the entire length.

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

Ultimate Strength Design Section Cuts:

Click **Sections** from the Capacity Check window 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

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.

Coordinates

The node points are defined as X and Y coordinates in units of inches. X coordinates run in ascending order from left to right. Y coordinates run in ascending order from top to bottom. These coordinates can be negative and the location of the 0,0 point is not critical.

Defaults

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 CWDEFAULT.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. Allowable stress coefficients and load cases can also be modified from the preset ACI values, if, for example, a different Code is used. If *Use Canadian material resistance factors* is selected, the Canadian Code, CSA A23.3 will be used for ultimate capacity determination.

Troubleshooting

- **Form printouts are hard to read:** From the **Print Setup** window, select **Options**. Under **Dithering**, select **Fine**. Then move the intensity control towards "Darker" if the printout is too light, or towards "Lighter" if it is too dark. Click "OK" to print.
- **Printed output is chopped off or turned 90 deg.**
From the **Print Setup** window, look at the **Paper Size** box. It should read "Letter 8 1/2 x 11 in". If not, click on the arrow to the right of the box to select the correct paper size. Also look at the **Orientation** box. The **Portrait** option should be selected.
- **"Error reading file LECWALL.LF" message**
The license file, LECWALL.LF, was not found and the program will not run. This file needs to be in the same directory as LECWALL.EXE. If the file has been lost or deleted, re-install LECWall by running setup.exe. Verify that the splash screen shows the correct version and latest revision date.

Frequently Asked Questions

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 LECCWall, 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).

Does LECCWall check if a shear connector can provide 100% composite action?

LECCWall displays the shear along the panel at 50 points. It is up to the engineer-user to determine how many shear connectors to use along the span to achieve the desired composite action. The individual connector shear strength could be obtained through testing or manufacturers' recommendations.

When checking handling I get a message in red text underneath the graphs that says "Note: Reinf ratio < ACI min. - capacity reduced 25%". What part of ACI does this reference?

This warning is for minimum flexure reinforcement. The relevant ACI Code section is 318-08, Section 10.5, specifically 10.5.3. 318-11 is the same. It states that A_s needs to be increased by 1/3 if minimum reinforcing for flexure is not provided. That is equivalent to a 25% reduction in capacity.

Also, please note that the reinforcing input unit is bar diameter, not area. If bar area is input by mistake, that could trigger the under-reinforced message.

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?

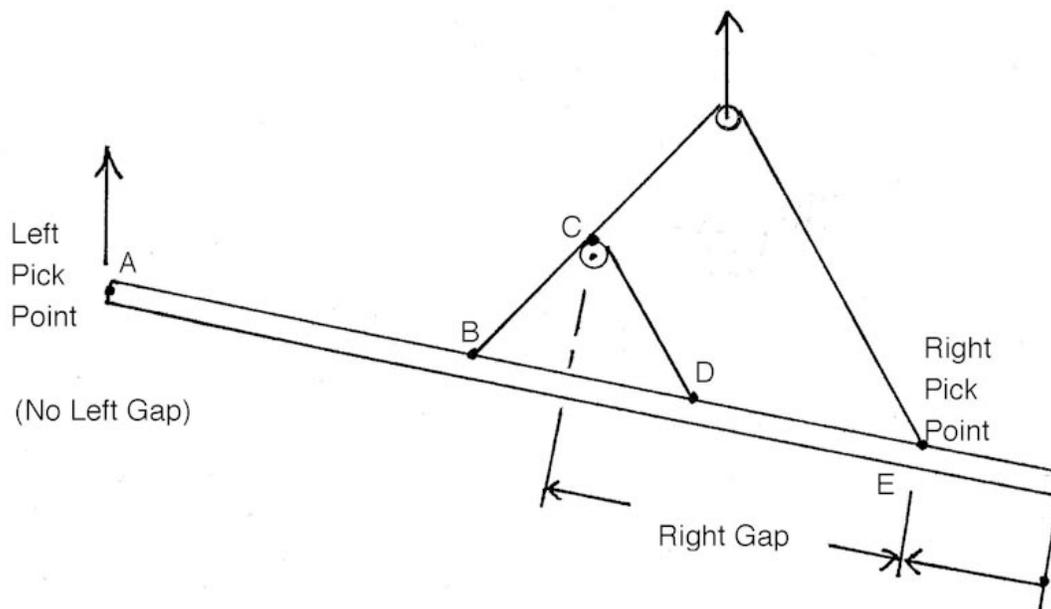
For service load cases, LECCWall uses the “Allowable Service Tension Stress Coefficient” specified on the “Load Case Coefficients” screen (attached). This defaults to $5 \sqrt{F'_c}$ for mild reinforced members. So, although the member is not cracked, tension stresses exceed that recommended by PCI for serviceability when exposed to view and weather (see PCI Handbook, 7th 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 1.2Mcr 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. The pick point locations and gaps for this case can be modeled as shown in the illustration below. The left pick point A is at the top of panel. The right pick point E is as shown, measured from the right edge (panel bottom). The right gap should be the distance from the left roller centerline C to E, not from the actual lifter locations, B and D. This is conservative, but necessary to correctly model the forces in the left and right lift lines.



LECWall gives me this message: "ACI 318-14 Maximum slenderness ratio of 1.4 may be exceeded."

This requirement can be found in Section 6.2.6 of ACI 318-14. The multistory (.W12) version of LECWall can't check this directly so it counts the number of P-Delta iterations instead. The message displays if iterations are greater than 4, signifying a slender member which may have significant secondary (P-Delta) moments.

There are two ways to check the member if you are receiving this message. One way is to calculate the moment at mid-height manually without any P-Delta effects and compare it to the LECWall moment. If the LECWall moment is less than 1.4 times the manual result, then it's OK. The second is to re-input the member as a single-story member (.WAL). This simpler analysis checks the 1.4 requirement directly and will give you a message if it is exceeded.

LOSCH SOFTWARE LICENSE AGREEMENT

April 10, 2016

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 30 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 the original disks and 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 the site addresses 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.

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. With respect to the physical diskette and physical documentation enclosed herein, Licensor warrants the same to be free of defects in materials and workmanship for a period of one year from the date of purchase. Licensor will provide periodic updates to the Software at its discretion and phone support for this same one year period.

5. CUSTOMER REMEDIES. In the event of notification within the warranty period of defects in material or workmanship, Licensor will replace the defective disk or documentation. The remedy for breach of this warranty shall be limited to replacement and shall not encompass any other damages, including but not limited to loss of profit, and special, incidental, consequential, or other similar claims.

6. 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.

7. 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 Wyoming.