

LECPres - Precast/Prestressed Beam and Slab Design

Contents

| | |
|--------|---|
| | <u>Introduction</u> |
| | <u>Methodology</u> |
| File: | <u>New Member File</u> <u>Open Member File</u> <u>Save Member File</u> <u>Save Member File As...</u> <u>Delete Member File</u> <u>Exit</u> |
| Input: | <u>Dimensions</u> <u>Section Properties</u> <u>Reinforcing</u> <u>Loadings</u> |
| Run: | <u>Run Problem</u> |
| Print: | <u>Print Preview</u> |
| Setup: | <u>Unlock</u> |
| Help: | <u>Help on Help</u> <u>Troubleshooting</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 beams and slabs. Some features are:

- Handles any combination of prestressing strand and/or mild reinforcing.
- Ultimate capacity based on strain compatibility
- Built-in torsion analysis
- Can model openings, pockets, partial ledges and other discontinuities along the member
- Straight and single or double strand depression
- Automatic or manual prestress loss calculations
- Camber prediction at release, erection and final, with and without live load
- Bi-linear deflection calculation is used when cracking stress is exceeded.
- Convenient built-in section property calculator.
- Transformed section is used for stress calculations.
- Two stage topping thickness option
- Supports sloped top flange
- Allows cantilevers at one or both ends.
- Provides top steel required for IT and L beams.
- Calculates horizontal shear reinforcement required at topping interface.
- Option for rebar anchorage at ends of member

- Multiple uniform, concentrated and trapezoidal loads, even concentrated moments. Any of these can be eccentric
- Can specify percent of live load to apply to service load stresses (useful for parking deck design).
- Support for the Canadian Code, as well as ACI 318-99 and ACI 318-08
- Choice of English or Metric units for input and output
- Easy to use Windows graphical interface with print preview.
- Compact three page printout lists all information.

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 beam or slab design calculations. To print it out, Click on Methodology in this paragraph, select **File** from the Help screen menu, then select **Print Topic**.

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

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

1. Select New Member File from the **File** pull-down menu. If another member file is loaded you will first be asked if you wish to save it.
2. Select the Dimensions tab from the **Input** 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. Click the **Save as Default** button to save certain input values as defaults for future new files.
3. Select the Section Properties tab from the **Input** menu. A form will appear listing concrete properties. Some boxes will already contain default values as a convenience. Click on Section Properties Calculator if desired to let the program calculate the section properties for you. The Special Composite Properties button is used to override the program's automatic composite property calculations and input special values manually.
4. Select the Reinforcing tab from the **Input** menu. There are three categories of reinforcing to fill in: strand, rebar and/or welded wire fabric.
5. Select the Loadings tab from the **Input** menu. Input the load information for each load and type.
6. At this point it would be a good idea to save your file. Select **File**, then Save Member File.
7. Select **Run** from the pull-down menu. The input will be processed and the output will be displayed as three pages in a **Print Preview** form. Click on **Diagrams** to view the ultimate load shear and moment diagram. If the output is acceptable, select **Print All** to send the three pages to the default Windows printer. If the member doesn't work, select **Close** and return to the input screen to make any modifications desired.

Methodology

Ultimate strength design and load factors are based on ACI 318-99, ACI 318-08 or CSA Standard A23.3, as appropriate.

All shear, moment, stress and deflection values are calculated at 40 equally spaced points along the member span.

Beam Analysis for Shear and moment at 40th Points:

The internal forces are determined by an approximate procedure. An equilibrium check is performed at 40 points along the span.

Shear and Torsion Capacity:

Shear strength is calculated according to ACI 318-08 Section 11.3. Shear reinforcement requirements are based on Section 11.4 and torsion is based on Section 11.5.

Moment Capacity:

Moment strength is based on strain compatibility and ACI 318-08 Section 18.7. The compression block can extend into the topping gap, the beam flange, the beam flange-to-web sloped portion and the beam web. Prestressing strand development length is based on Section 12.9.

Stresses:

End release stresses are calculated at the point of prestress transfer equal to 50 strand diameters from the end of the member. Allowable release and service load stresses are based on ACI 318-08 Section 18.4, with the exception of allowable service load tension stress, which is input by the user. Prestress losses are calculated using Section 5.7 of the PCI Design Handbook, Seventh Edition.

Deflections:

The moment-area method is used to calculate flexural displacements. When the bottom tension stress exceeds $7.5 * \text{SQRT}(F'c)$ bi-linear behavior is assumed, based on the PCI Handbook Section 5.8.3.

References:

- **Building Code Requirements for Structural Concrete (ACI 318-08)**, American Concrete Institute, Detroit, MI, 2008.
- **PCI Design Handbook, Seventh Edition**, Precast/Prestressed Concrete Institute, Chicago, IL, 2010.

New Member File

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

Open Member File

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

Save Member File

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

Save Member File As...

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

Delete Member 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 member file and exit to Windows.

Dimensions

The **Dimension** tab lists the member cross-section dimension inputs. Use the Tab key on the keyboard to cycle through and highlight each input variable to edit. Topping dimensions should be left at zero if there is no topping. Topping gap width and gap thickness should be zero if there is no gap in the topping. For members without a separate flange, such as rectangular, L or inverted Tee beams, the input for flange thickness should be left at zero. (Specifying the member height as the flange thickness will also work, but it is easier and more fool-proof to leave the value at zero.)

The following values will be saved as defaults for new files if the **Save as Default** button is clicked:

Project Name Designer's Initials Number of Webs

Click **Cancel** to discard any changes and close the Input form. Click **Done** to keep changes and close the Input form. Note: The changed input values are not saved to disk until Save Member File is selected from the **File** menu.

Section Properties

The **Section Properties** tab lists the member section and concrete property inputs. Use the Tab key on the keyboard to cycle through and highlight each input variable to edit. These properties are non-composite, non-transformed properties. The program will calculate composite and transformed properties automatically.

The value for outside perimeter of section (P_{cp}) is used both for torsion design and for prestress loss calculation. If there is no significant torsion, leave A_0h and P_h as zero to bypass the torsion routines. If P_{cp} is left at zero, the perimeter will be calculated automatically from the other input values. Note: If the member is a ledger or inverted-tee beam the P_{cp} value should be always be input manually, since the automatic calculation will be inaccurate because of the ledge.

Modulus of elasticity will also be calculated automatically if left at zero, as noted on the screen.

The Section Properties Calculator is provided as a convenience for finding the area, Y_b and moment of inertia of the section.

Click Special Composite Properties to override the automatic calculations and manually input the composite properties.

Click Special Section Properties to allow the modelling of discontinuities in the member, such as openings, notches, pockets, partial-length ledges, etc.

The following values will be saved as defaults for new files if the **Save as Default** button is clicked:

Concrete Weight Initial F'c Final F'c Initial E
Final E Allowable Service Tension Coefficient % Relative Humidity.

Click **Cancel** to discard any changes and close the Input form. Click **Done** to keep changes and close the Input form. Note: The changed input values are not saved to disk until Save Member File is selected from the **File** menu.

Section Properties Calculator

The Section Properties Calculator is provided as a convenience for finding the area, Y_b and moment of inertia of the section.

The program sums up a sequence of trapezoidal areas to arrive at the property values. Each node corresponds to the width and elevation of a horizontal face of a trapezoid. Use the total section width for multi-stem members. The following is an example using a standard 10DT32 section from the PCI Handbook, 4th Ed, pg 2-23:

| Node | X | Y |
|------|------|----|
| 1 | 9.5 | 0 |
| 2 | 15.5 | 30 |
| 3 | 120 | 30 |
| 4 | 120 | 32 |

Click on a cell, or use the arrow keys to highlight a cell and press Enter to alter the value. Compare the output to the values in the PCI table as a check (they should match exactly):

Area = 615 in² Y_b = 21.98 in. I = 59,720 in⁴

Special Composite Properties

Click Special Composite Properties to override the automatic calculations and manually input the composite properties. This feature can be useful when the topping configuration does not match any of the configurations supported by the program. The values that can be specified are:

| | | |
|--------------------|-------|-------|
| Moment of Inertia: | I | |
| Neutral Axis: | Y_b | Y_t |
| Section Modulus: | Z_b | Z_t |

Note: **All** properties must be input if this option is taken. Also note that these values are not saved to disk. They are discarded when a new file is loaded or when the program exits.

Special Section Properties

Click Special Section Properties from the Section Properties tab of the Input form to access this powerful tool. It allows the modelling of discontinuities in the member. Some examples include partial-length ledges, flange or web openings, notches and pockets. The section properties input for cases 1 through 6 replace the full-length values for the location and length specified along the member.

Be sure to use the strand debonding inputs to simulate cut strands at openings. Also adjust rebar start locations and lengths to avoid openings. **THIS WILL NOT BE DONE AUTOMATICALLY.**

The **Height of web opening** input will subtract this height times the beam width from the shear area for shear calculations. **IT IS ASSUMED THAT THE OPENING WILL REMAIN OUTSIDE OF THE**

COMPRESSION BLOCK. AN OPENING AT THE VERY TOP OR BOTTOM OF THE MEMBER IS NOT PERMITTED.

If there is a notch on the bottom of the beam, input the **Start of section from beam bottom**, which is also the height of the notch. This will ensure that strand and bar locations are correct within the special section.

Note: **All** listed properties must be input for each case where Special Section length is greater than zero.

Reinforcing

The **Reinforcing** tab lists the strand and rebar properties. Use the Tab key on the keyboard to cycle through and highlight each input variable to edit. Strand initial and final loss percent will be automatically calculated if the inputs are left at zero. The bonded and unbonded development factor inputs allow a multiplier to be used for strand development length. A multiplier of 2 is commonly used for strand that are unbonded at some point along the member. This value may not always be appropriate. In some cases, due to strain compatibility, unbonded strands do not contribute at all to the member's strength. The development of unbonded tendons is a complex topic which is addressed in the following article:

"Strength of Prestressed Concrete Members at Sections Where Strands are not Fully Developed", by Leslie D. Martin and Walter J. Korkosz, PCI Journal, Sept/Oct 1995, Pg 58.

Click on **Strand Locations** and **Rebar Locations** to locate the reinforcing. Click on a cell, or use the arrow keys to highlight a cell and press Enter to alter the value. If the strand is not depressed, leave the input at zero for depression start and end. Yb Center should also equal Yb End. If depressed at the center only, input zero for depression start and end. The program will find the member center and depress the strand at that point.

The following values will be saved as defaults for new files if the **Save as Default** button is clicked:

| | | |
|-----------------------|----------------------|---------------|
| Strand Diameter | Strand Area | Strand Fpu |
| Bonded Dev. Factor | UnBonded Dev. Factor | |
| Strand Initial Loss % | Strand Final Loss % | Low-Lax (Y/N) |
| Stirrup Fy | Longitudinal Fy | |

Click **Cancel** to discard any changes and close the Input form. Click **Done** to keep changes and close the Input form. Note: The changed input values are not saved to disk until Save Member File is selected from the **File** menu.

Loadings

The **Loadings** tab lists the member loading information. Use the Tab key on the keyboard to cycle through and highlight each input variable to edit. Up to 10 loadings for each type of load are supported. click the Edit box to input each loading. Click on a cell, or use the arrow keys to highlight a cell and press Enter to alter the value.

Note: Do not include the member or topping weight as dead load. Member and topping weight is automatically calculated and included by the program. The percent of dead load applied before composite defaults to zero. In this case, the member is assumed to be fully composite before any superimposed dead load (other than the topping) is applied. The member service stresses and deflections would be affected by this value.

The percent of live load applied to service load defaults to 100. This value should normally stay at 100 percent. An exception to this rule could be the roof of a parking deck or balcony where floor live load and roof snow load are applied simultaneously to determine ultimate strength. The designer may feel that these loads need not be applied together for the stress and deflection checks and use a value less than 100%.

The load eccentricity value is used to determine the torsional moment along the member. This value can be negative if the load is on the opposite side of the member from the other loads.

Positive concentrated moments act clockwise, while negative moments act counter-clockwise.

Click **Cancel** to discard any changes and close the Input form. Click **Done** to keep changes and close the Input form. Note: The changed input values are not saved to disk until Save Member File is selected from the **File** menu.

Run Problem

Select **Run** from the pull-down menu. The input will be processed and the output will be displayed as three or four pages in a **Print Preview** form. Click on **Diagrams** to view the ultimate load shear and moment diagram. If the output is acceptable, select **Print All** to send the three pages to the default Windows printer. If the member doesn't work, select **Close** and return to the input screen to make any modifications desired.

Print Preview

The output pages will be displayed in a **Print Preview** window. Click on **Diagrams** to view the ultimate load shear and moment diagram. If the output is acceptable, select **Print All** to send the three or four pages to the default Windows printer. If the member doesn't work, select **Close** and return to the input screen to make any modifications desired.

Unlock

The trial version has a built-in timer which automatically "shuts off" the program after the trial period has expired. The trial period can be optionally extended by contacting Losch Software.

The licensed version does not have a timer and this screen is disabled.

Help on Help

Press **F1** at any time to show this *Help* screen. Clicking on any topics that are shown as green will cause an instant jump to that topic. Use the **Contents** button to return to the Table of Contents. The **Back** button will take you back to the previous screen. Select **File/Exit** from the pull-down menu to exit *Help*.

Troubleshooting

- **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.

- **I installed an update to the program, but the revision date did not change when I ran it.** The program needs to be un-installed before installing the update. Click on Start, Settings, Control Panel, Add/Remove Programs. Highlight "Losch Software Precast Prestressed Beam Design" and click the Add/Remove button. It's OK, member input files will not be erased.

License Agreement

LOSCH SOFTWARE LICENSE AGREEMENT

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

1A. TRIAL LICENSE GRANT. Losch Software, LLC (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. SITE 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.

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 Nevada.