pcaSlab

User's Manual

pcaSlab 1.10

CONCRETE SOFTWARE SOLUTIONS

August 2004

Table of Contents

Legal and Contact Information

Copyright Information

Copyright Notice

Trademarks

Evaluation Software License Agreement

PORTLAND CEMENT ASSOCIATION
EVALUATION SOFTWARE LICENSE AGREEMENT

BY CLICKING THE ACCEPTED ICON BELOW, OR BY INSTALLING, COPYING, OR OTHERWISE USING THE SOFTWARE OR USER DOCUMENTATION, YOU AGREE TO BE BOUND BY THE TERMS OF THIS AGREEMENT, INCLUDING, BUT NOT LIMITED TO, THE WARRANTY DISCLAIMERS, LIMITATIONS OF LIABILITY AND TERMINA TION PROVISIONS BELOW. IF YOU DO NOT AGREE TO THE TERMS OF THIS AGREEMENT, DO NOT INSTALL OR USE THE SOFTWARE OR USER DOCUMENTA TION, EXIT THIS APPLICATION NOW AND RETURN THE SOFTWARE AND USER DOCUMENTATION TO PCA.

PORTLAND CEMENT ASSOCIATION, 5420 OLD ORCHARD ROAD, SKOKIE, ILLI NOIS 60077 (HEREAFTER PCA), GRANTS THE CUSTOMER A PERSONAL, NONEX CLUSIVE, LIMITED, NONTRANSFERABLE LICENSE TO USE THIS SOFTWARE AND USER DOCUMEN TA TION SOLELY FOR TRIAL AND EVALUATION PURPOSES ONLY IN ACCORDANCE WITH THE TERMS AND CONDITIONS OF THIS AGREEMENT. SOFT WARE AND USER DOCUMENTATION IS SUPPLIED TO CUSTOMER EITHER BY PCA DIRECTLY OR THROUGH AN AUTHORIZED DEALER OF PCA (HEREAFTER DEALER).

WHILE PCA HAS TAKEN PRECAUTIONS TO ASSURE THE CORRECTNESS OF THE ANALYTICAL SOLUTION AND DESIGN TECHNIQUES USED IN THIS SOFTWARE, IT CANNOT AND DOES NOT GUARANTEE ITS PERFORMANCE, NOR CAN IT OR DOES IT BEAR ANY RESPONSIBILITY FOR DEFECTS OR FAILURES IN STRUCTURES IN CONNECTION WITH WHICH THIS SOFTWARE MAY BE USED. DEALER (IF ANY) HAS NOT PARTICIPATED IN THE DESIGN OR DEVELOPMENT OF THIS SOFTWARE AND NEITHER GUARANTEES THE PERFORMANCE OF THE SOFTWARE NOR BEARS ANY RESPONSIBILITY FOR DEFECTS OR FAILURES IN STRUCTURES IN CONNEC TION WITH WHICH THIS SOFTWARE IS USED.

PCA AND DEALER (IF ANY) EXPRESSLY DISCLAIM ANY WARRANTY THAT: (A) THE FUNCTIONS CONTAINED IN THE SOFTWARE WILL MEET THE REQUIRE MENTS OF CUSTOMER OR OPERATE IN COMBINATIONS THAT MAY BE SE LECTED FOR USE BY CUSTOMER; (B) THE OPERATION OF THE SOFTWARE WILL BE FREE OF ALL "BUGS" OR PROGRAM ERRORS; OR (C) THE SOFTWARE CON FORMS TO ANY PERFORMANCE SPECIFICATIONS. CUSTOMER ACKNOWLEDGES THAT PCA IS UNDER NO OBLIGATION TO PROVIDE ANY SUPPORT, UPDATES, BUG FIXES OR ERROR CORRECTIONS TO OR FOR THE SOFTWARE OR USER DOCUMENTATION.

THE LIMITED WARRANTIES IN SECTION 6 HEREOF ARE IN LIEU OF ALL OTHER WARRANTIES, EXPRESS OR IMPLIED, INCLUDING, BUT NOT LIMITED TO, ANY IMPLIED WARRANTIES OF NON-INFRINGEMENT, MERCHANTABILITY OR FIT NESS FOR A PARTICULAR PURPOSE, EACH OF WHICH IS HEREBY DIS CLAIMED. EXCEPT AS SET FORTH IN SECTION 6, THE SOFTWARE AND USER DOCUMENTA TION ARE PROVIDED ON AN "AS-IS" BASIS.

IN NO EVENT SHALL PCA OR DEALER (IF ANY) BE LIABLE FOR: (A) LOSS OF PROFITS, DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, PUNITIVE, CONSE QUEN TIAL OR OTHER DAMAGES, EVEN IF PCA OR DEALER (IF ANY) HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES; (B) ANY CLAIM AGAINST CUSTOMER BY ANY THIRD PARTY; OR (C) ANY DA MAGES CAUSED BY (1) DELAY IN DELIVERY OF THE SOFT WARE OR USER DOCUMEN TATION UNDER THIS AGREEMENT; (2) THE PERFORMANCE OR NON PERFORMANCE OF THE SOFTWARE; (3) RESULTS FROM USE OF THE SOFTWARE OR USER DOCU MENTA TION, INCLUDING, WITHOUT LIMI TA TION, MISTAKES, ERRORS, INAC CURACIES, FAILURES OR CUSTOMER'S INABILITY TO PROVIDE SERVICES TO THIRD PARTIES THROUGH USE OF THE SOFTWARE OR USER DOCUMENTA TION; (4) CUSTOMER'S FAILURE TO PERFORM CUSTOMER'S RESPON SIBILI TIES; (5) PCA NOT PROVIDING UPDATES, BUG FIXES OR CORRECTIONS TO OR FOR ANY OF THE SOFTWARE OR USER DOCUMENTATION; (6) LABOR, EX PENSE OR MATERIALS NECESSARY TO REPAIR DAMAGE TO THE SOFTWARE OR USER DOCUMENTATION CAUSED BY (a) ACCI DENT, (b) NEGLI GENCE OR ABUSE BY CUSTOMER, (c) ACTS OF THIRD PERSONS INCLUDING, BUT NOT LIMITED TO, INSTALLATION, REPAIR, MAINTENANCE OR OTHER CORREC TIVE WORK RELATED TO ANY EQUIPMENT BEING USED, (d) CAUSES EXTER NAL TO THE SOFTWARE SUCH AS POWER FLUCTUATION AND FAILURES, OR (e) FLOODS, WINDSTORMS OR OTHER ACTS OF GOD. MOREOVER, IN NO EVENT SHALL PCA BE LIABLE FOR WARRANTIES, GUARANTEES, REPRE SEN TA TIONS OR ANY OTHER UNDERSTAN DINGS BETWEEN CUSTOMER AND DEALER (IF ANY) RELATING TO THE SOFT WARE OR USER DOCUMENTATION.

THIS AGREEMENT CONSTITUTES THE ENTIRE AND EXCLUSIVE AGREEMENT BETWEEN CUSTOMER AND PCA AND DEALER (IF ANY) WITH RESPECT TO THE SOFTWARE AND USER DOCUMENTATION TO BE FURNISHED HEREUNDER. IT IS A FINAL EXPRESSION OF THAT AGREEMENT AND UNDERSTANDING. IT SUPER SEDES ALL PRIOR COMMUNICATIONS BETWEEN THE PARTIES (INCLUDING ANY EVALUATION LICENSE AND ALL ORAL AND WRITTEN PROPOSALS). ORAL STATEMENTS MADE BY PCA'S OR DEALER'S (IF ANY) REPRESENTATIVES ABOUT THE SOFTWARE OR USER DOCU MEN TATION DO NOT CONSTITUTE REPRESEN TA TIONS OR WARRANTIES, SHALL NOT BE RELIED ON BY CUSTOMER, AND ARE NOT PART OF THIS AGREEMENT.

1. LICENSE RESTRICTIONS

(a) Except as expressly provided in this Agreement or as otherwise authorized in writing by PCA, Customer has no right to: (1) use, print, copy, display, reverse assemble, reverse engineer, translate or decompile the Software or User Documentation in whole or in part; (2) disclose, publish, release, sublicense or transfer to another person any Software or User Documentation; (3) reproduce the Software or User Documentation for the use or benefit of anyone other than Customer; or (4) modify any Software or User Documentation. All rights to the Software and User Documentation not expressly granted to Customer hereunder are retained by PCA. All copyrights and other proprietary rights except as expressed elsewhere in the Software or User Documentation and legal title thereto shall remain in PCA. Customer may use the Software at only one designated workstation at Customer's site at any given time. Customer may not transmit the Software electronically to any other workstation, computer, node or terminal device whether via a local area network, a wide area network, telecommunications transmission, the Internet or other means now known or hereafter created without prior written permission by PCA.

(b) Customer acknowledges that this is a limited license for trial and evaluation purposes only. This limited license shall automatically terminate upon the earlier of: (1) ten executions of the Software on the computer on which it is installed; or (2) fifteen days after the installation of the Software. Thereafter, Customer may only use the Software and Documentation if it acquires a production license for the same.

2. TERM AND TERMINATION

(a) This Agreement shall be in effect from the date Customer clicks the accepted icon below or installs, copies or otherwise uses the Software or User Documentation until: (1) it is terminated by Customer, by Dealer (if any) on behalf of Customer or PCA or by PCA as set forth herein; or (2) the limited trial and evaluation license terminates.

(b) This Agreement may be terminated by PCA without cause upon 30 days' written notice or immediately upon notice to Customer if Customer breaches this Agreement or fails to comply with any of its terms or conditions. This Agreement may be terminated by Customer without cause at any time upon written notice to PCA.

3. BACKUP AND REPLACEMENT COPIES

Customer may make one copy of the Software for back-up and archival purposes only, provided PCA's copyright and other proprietary rights notices are included on such copy.

4. PROTECTION AND SECURITY

(a) Customer shall not provide or otherwise make available any of the Software or User Documentation in any form to any person other than employees of Customer with the need to know, without PCA's written permission.

(b) All Software and User Documentation in Customer's possession including, with out limitation, translations, compilations, back-up, and partial copies is the property of PCA. Upon termination of this Agreement for any reason, Customer shall immediately destroy all Software and User Documentation, including all media, and destroy any Software that has been copied onto other magnetic storage devices. Upon PCA's request, Customer shall certify its compliance in writing with the foregoing to PCA.

(c) Customer shall take appropriate action, by instruction, agreement or otherwise, with any persons permitted access to the Software or User Documentation, to enable Customer to satisfy its obligations under this Agreement with respect to use, copying, protection, and security of the same.

(d) If PCA prevails in an action against Customer for breach of the provisions of this Section 4, Customer shall pay the reasonable attorneys' fees, costs, and expenses incurred by PCA in connection with such action in addition to any award of damages.

5. CUSTOMER'S RESPONSIBILITIES

The essential purpose of this Agreement is to provide Customer with limited use rights to the Software and User Documentation. Customer accepts full responsibility for: (a) selection of the Software and User Documentation to satisfy Customer's business needs and achieve Customer's intended results; (b) the use, set-up and installation of the Software and User Documentation; (c) all results obtained from use of the Software and User Documentation; and (d) the selection, use of, and results obtained from any other software, programming equipment or services used with the Software or User Documentation.

6. LIMITED WARRANTIES

PCA and Dealer (if any) warrants to Customer that: (a) PCA and Dealer (if any) has title to the Software and User Documentation and/or the right to grant Customer the rights granted hereunder; (b) the Software and User Documentation provided hereunder is PCA's most current production version thereof; and (c) the copy of the Software provided hereunder is an accurate reproduction of the original from which it was made.

7. LIMITATION OF REMEDY

(a) PCA AND DEALER (IF ANY) HAS NO LIABILITY UNDER THIS AGREEMENT. CUSTOMER'S EX CLUSIVE REMEDY FOR DAMAGES DUE TO PERFORMANCE OR NONPER FOR MANCE OF ANY SOFTWARE OR USER DOCU MENTATION, PCA, DEALER (IF ANY), OR ANY OTHER CAUSE WHATSO EVER, AND REGARDLESS OF THE FORM OF ACTION, WHETHER IN CONTRACT OR IN TORT, INCLUDING NEGLIGENCE, SHALL BE LIMITED TO CUSTOMER STOP PING ALL USE OF THE SOFTWARE AND USER DOCUMENTATION AND RETUR NING THE SAME TO PCA.

(b) NEITHER PCA NOR DEALER (IF ANY) IS AN INSURER WITH RE GARD TO THE PERFORMANCE OF THE SOFTWARE OR USER DOCUMENTA TION. THE TERMS OF THIS AGREEMENT, INCLUDING, BUT NOT LIMITED TO, THE LIMITED WARRANTIES, AND THE LIMITATION OF LIABILITY AND RE MEDY, ARE A REFLEC TION OF THE RISKS ASSUMED BY THE PARTIES. IN ORDER TO OBTAIN THE SOFTWARE AND USER DOCUMENTATION FROM PCA OR DEALER (IF ANY), CUSTOMER HEREBY ASSUMES THE RISKS FOR (1) ALL LIABILITIES DISCLAIMED BY PCA AND DEALER (IF ANY) ON THE FACE HEREOF; AND (2) ALL ACTUAL OR ALLEGED DAMAGES IN CONNECTION WITH THE USE OF THE SOFTWARE AND USER DOCUMENTATION. THE ESSENTIAL PURPOSE OF THE LIMITED REMEDY PROVIDED CUSTOMER HEREUNDER IS TO ALLOCATE THE RISKS AS PROVIDED ABOVE.

8. U.S. GOVERNMENT RESTRICTED RIGHTS

This commercial computer software and commercial computer software documentation were developed exclusively at private expense by Portland Cement Association, 5420 Old Orchard Road, Skokie, Illinois, 60077. U.S. Government rights to use, modify, release, reproduce, perform, display or disclose this computer soft ware and computer software documentation are subject to the restrictions of DFARS 227.7202-1(a) (June 1995) and DFARS 227.7202-3(a) (June 1995), or the Restricted Rights provisions of FAR 52.227-14 (June 1987) and FAR 52.227-19 (June 1987), as applicable.

9. GENERAL

(a) No action arising out of any claimed breach of this Agreement or transactions under this Agreement may be brought by Customer more than two years after the cause of such action has arisen.

(b) Customer may not assign, sell, sublicense or otherwise transfer this Agreement, the license granted herein or the Software or User Documentation by operation of law or other wise without the prior written consent of PCA. Any attempt to do any of the foregoing without PCA's consent is void.

(c) Customer acknowledges that the Software, User Documentation and other proprietary information and materials of PCA are unique and that, if Customer breaches this Agreement, PCA may not have an adequate remedy at law and PCA may enforce its rights hereunder by an action for damages and/or injunctive or other equitable relief without the necessity of proving actual damage or posting a bond therefor.

(D) THE RIGHTS AND OBLIGATIONS UNDER THIS AGREEMENT SHALL NOT BE GOVERNED BY THE UNITED NATIONS CONVENTION ON CONTRACTS FOR THE INTERNATIONAL SALE OF GOODS, THE APPLICATION OF WHICH IS EX PRESSLY EXCLUDED, BUT SUCH RIGHTS AND OBLIGATIONS SHALL INSTEAD BE GOVERNED BY THE LAWS OF THE STATE OF ILLINOIS, APPLICABLE TO CON TRACTS ENTERED INTO AND PERFORMED ENTIRELY WITHIN THE STATE OF ILLINOIS AND APPLICABLE FEDERAL (U.S.) LAWS. UCITA SHALL NOT APPLY TO THIS AGREEMENT.

(E) THIS AGREEMENT SHALL BE TREATED AS THOUGH IT WERE EXE CUTED IN THE COUNTY OF COOK, STATE OF ILLINOIS, AND WAS TO HAVE BEEN PERFORMED IN THE COUNTY OF COOK, STATE OF ILLINOIS. ANY ACTION RE LATING TO THIS AGREEMENT SHALL BE INSTITUTED AND PROSECUTED IN A COURT LOCATED IN COOK COUNTY, ILLINOIS. CUSTOMER SPECIFICALLY CON SENTS TO EXTRATERRITORIAL SERVICE OF PROCESS.

(f) Except as prohibited elsewhere in this Agreement, this Agreement shall be binding upon and inure to the benefit of the personal and legal representatives, permitted successors, and permitted assigns of the parties hereto.

(g) All notices, demands, consents or requests that may be or are required to be given by any party to another party shall be in writing. All notices, demands, consents or requests given by the parties hereto shall be sent either by U.S. certified mail, postage prepaid or by an overnight international delivery service, addressed to the respective parties. Notices, demands, consents or requests served as set forth herein shall be deemed sufficiently served or given at the time of receipt thereof.

(h) The various rights, options, elections, powers, and remedies of a party or parties to this Agreement shall be construed as cumulative and no one of them exclusive of any others or of any other legal or equitable remedy that said party or parties might otherwise have in the event of breach or default in the terms hereof. The exercise of one right or remedy by a party or parties shall not in any way impair its rights to any other right or remedy until all obligations imposed on a party or parties have been fully performed.

(i) No waiver by Customer, PCA or Dealer (if any) of any breach, provision, or de fault by the other shall be deemed a waiver of any other breach, provision or default.

(j) The parties hereto, and each of them, agree that the terms of this Agreement shall be given a neutral interpretation and any ambiguity or uncertainty herein should not be construed against any party hereto.

(k) If any provision of this Agreement or portion thereof is held to be unenforceable or invalid by any court or competent jurisdiction, such decision shall not have the effect of invalidating or voiding the remainder of this Agreement, it being the intent and agreement of the parties that this Agreement shall be deemed amended by modifying such provision to the extent necessary to render it enforceable and valid while preserving its intent or, if such modification is not possible, by substituting therefor another provision that is enforceable and valid so as to materially effectuate the parties' intent.

(l) Except as set forth herein, this Agreement may be modified or amended only by a written instrument signed by a duly authorized representative of PCA and Customer.

Software License Agreement

PORTLAND CEMENT ASSOCIATION
SOFTWARE LICENSE AGREEMENT

BY CLICKING THE ACCEPTED ICON BELOW, OR BY INSTALLING, COPYING, OR OTHERWISE USING THE SOFTWARE OR USER DOCUMENTATION, YOU AGREE TO BE BOUND BY THE TERMS OF THIS AGREEMENT, INCLUDING, BUT NOT LIMITED TO, THE WARRANTY DISCLAIMERS, LIMITATIONS OF LIABILITY AND TERMINA TION PROVISIONS BELOW. IF YOU DO NOT AGREE TO THE TERMS OF THIS AGREEMENT, DO NOT INSTALL OR USE THE SOFTWARE OR USER DOCUMENTA TION, EXIT THIS APPLICATION NOW AND RETURN THE SOFTWARE AND USER DOCUMENTATION TO PCA FOR A FULL REFUND WITHIN THIRTY DAYS AFTER YOUR RECEIPT OF THE SOFTWARE AND USER DOCUMENTATION.

PORTLAND CEMENT ASSOCIATION, 5420 OLD ORCHARD ROAD, SKOKIE, ILLINOIS 60077 (HEREAFTER PCA), GRANTS THE CUSTOMER A PERSONAL, NONEXCLUSIVE, LIMITED, NONTRANSFERABLE LICENSE TO USE THIS SOFT WARE AND USER DOCUMEN TA TION IN ACCORDANCE WITH THE TERMS AND CONDITIONS OF THIS AGREEMENT. SOFTWARE AND USER DOCUMEN TA TION IS SUPPLIED TO CUSTO MER EITHER BY PCA DIRECTLY OR THROUGH AN AUTHO RIZED DEALER OF PCA (HEREAFTER DEALER).

WHILE PCA HAS TAKEN PRECAUTIONS TO ASSURE THE CORRECTNESS OF THE ANALYTICAL SOLUTION AND DESIGN TECHNIQUES USED IN THIS SOFTWARE, IT CANNOT AND DOES NOT GUARANTEE ITS PERFORMANCE, NOR CAN IT OR DOES IT BEAR ANY RESPONSIBILITY FOR DEFECTS OR FAILURES IN STRUCTURES IN CONNECTION WITH WHICH THIS SOFTWARE IS USED. DEALER (IF ANY) HAS NOT PARTICIPATED IN THE DESIGN OR DEVELOPMENT OF THIS SOFTWARE AND NEITHER GUARANTEES THE PERFORMANCE OF THE SOFTWARE NOR BEARS ANY RESPONSIBILITY FOR DEFECTS OR FAILURES IN STRUCTURES IN CONNECTION WITH WHICH THIS SOFTWARE IS USED.

PCA AND DEALER (IF ANY) EXPRESSLY DISCLAIM ANY WARRANTY THAT: (A) THE FUNCTIONS CONTAINED IN THE SOFTWARE WILL MEET THE REQUIREMENTS OF CUSTOMER OR OPERATE IN COMBINATIONS THAT MAY BE SELECTED FOR USE BY CUSTOMER; (B) THE OPERATION OF THE SOFTWARE WILL BE FREE OF ALL "BUGS" OR PROGRAM ERRORS; OR (C) THE SOFTWARE CONFORMS TO ANY PERFORMANCE SPECIFICATIONS. CUSTOMER ACKNOWLEDGES THAT PCA IS UNDER NO OBLIGATION TO PROVIDE ANY SUPPORT, UPDATES, BUG FIXES OR ERROR CORRECTIONS TO OR FOR THE SOFTWARE OR USER DOCUMENTATION.

THE LIMITED WARRANTIES IN SECTION 7 HEREOF ARE IN LIEU OF ALL OTHER WARRANTIES, EXPRESS OR IMPLIED, INCLUDING, BUT NOT LIMITED TO, ANY IMPLIED WARRANTIES OF NON-INFRINGEMENT, MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE, EACH OF WHICH IS HEREBY DISCLAIMED. EXCEPT AS SET FORTH IN SECTION 7, THE SOFTWARE AND USER DOCUMENTATION ARE PROVIDED ON AN "AS-IS" BASIS.

IN NO EVENT SHALL PCA OR DEALER (IF ANY) BE LIABLE FOR: (A) LOSS OF PROFITS, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, PUNITIVE, CONSE QUEN TIAL OR OTHER DAMAGES, EVEN IF PCA OR DEALER (IF ANY) HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES; (B) ANY CLAIM AGAINST CUSTOMER BY ANY THIRD PARTY EXCEPT AS PROVIDED IN SECTION 8 ENTITLED "INFRINGE MENT"; OR (C) ANY DAMAGES CAUSED BY (1) DELAY IN DELIVERY OF THE SOFT WARE OR USER DOCUMEN TATION UNDER THIS AGREEMENT; (2) THE PERFORMANCE OR NONPERFORMANCE OF THE SOFTWARE; (3) RESULTS FROM USE OF THE SOFTWARE OR USER DOCUMEN TATION, INCLUDING, WITHOUT LIMI TA TION, MISTAKES, ERRORS, INACCU RACIES, FAILURES OR CUSTOMER'S INABILITY TO PROVIDE SERVICES TO THIRD PARTIES THROUGH USE OF THE SOFTWARE OR USER DOCUMENTA TION; (4) CUSTOMER'S FAILURE TO PERFORM CUSTOMER'S RESPONSIBILI TIES; (5) PCA NOT PROVIDING UPDATES, BUG FIXES OR CORRECTIONS TO OR FOR ANY OF THE SOFTWARE OR USER DOCUMENTATION; (6) LABOR, EX PENSE OR MATERIALS NECESSARY TO REPAIR DAMAGE TO THE SOFTWARE OR USER DOCUMENTATION CAUSED BY (a) ACCIDENT, (b) NEGLI GENCE OR ABUSE BY CUSTOMER, (c) ACTS OF THIRD PERSONS INCLUDING, BUT NOT LIMITED TO, INSTALLATION, REPAIR, MAINTENANCE OR OTHER CORREC TIVE WORK RELATED TO ANY EQUIPMENT BEING USED, (d) CAUSES EXTER NAL TO THE SOFTWARE SUCH AS POWER FLUCTUATION AND FAILURES, OR (e) FLOODS, WINDSTORMS OR OTHER ACTS OF GOD. MOREOVER, IN NO EVENT SHALL PCA BE LIABLE FOR WARRANTIES, GUARANTEES, REPRESEN TA TIONS OR ANY OTHER UNDERSTANDINGS BETWEEN CUSTOMER AND DEALER (IF ANY) RELATING TO THE SOFTWARE OR USER DOCUMENTATION.

THIS AGREEMENT CONSTITUTES THE ENTIRE AND EXCLUSIVE AGREEMENT BETWEEN CUSTOMER AND PCA AND DEALER (IF ANY) WITH RESPECT TO THE SOFTWARE AND USER DOCUMENTATION TO BE FURNISHED HEREUNDER. IT IS A FINAL EXPRESSION OF THAT AGREEMENT AND UNDERSTANDING. IT SUPER SEDES ALL PRIOR COMMUNICATIONS BETWEEN THE PARTIES (INCLUDING ANY EVALUATION LICENSE AND ALL ORAL AND WRITTEN PROPOSALS). ORAL STATEMENTS MADE BY PCA'S OR DEALER'S (IF ANY) REPRESENTATIVES ABOUT THE SOFTWARE OR USER DOCU MEN TATION DO NOT CONSTITUTE REPRESEN TA TIONS OR WARRANTIES, SHALL NOT BE RELIED ON BY CUSTOMER, AND ARE NOT PART OF THIS AGREEMENT.

1. LICENSE RESTRICTIONS

(a) Except as expressly provided in this Agreement or as otherwise authorized in writing by PCA, Customer has no right to: (1) use, print, copy, display, reverse assemble, reverse engineer, translate or decompile the Software or User Documentation in whole or in part; (2) disclose, publish, release, sublicense or transfer to another person any Software or User Documentation; (3) reproduce the Software or User Documentation for the use or benefit of anyone other than Customer; or (4) modify any Software or User Documentation. All rights to the Software and User Documentation not expressly granted to Customer hereunder are retained by PCA. All copyrights and other proprietary rights except as expressed elsewhere in the Software or User Documentation and legal title thereto shall remain in PCA. Customer may use the Software at only one designated workstation at Customer's site at any given time. Customer may not transmit the Software electronically to any other workstation, computer, node or terminal device whether via a local area network, a wide area network, telecommunications transmission, the Internet or other means now known or hereafter created without prior written permission by PCA.

(B) CUSTOMER ACKNOWLEDGES THAT THE REGISTRATION PRO CESS FOR THE SOFTWARE RESULTS IN THE GENERATION OF A UNIQUE LICENSE KEY AND ACTIVATION CODE PAIR. ONCE THE LICENSE KEY AND ACTIVATION CODE ARE ENTERED DURING THE INSTALLATION PROCESS, THE SOFTWARE WILL ONLY WORK ON THE COMPUTER ON WHICH THE SOFT WARE IS INITIALLY INSTALLED. IF YOU NEED TO DEINSTALL THE SOFTWARE AND REINSTALL THE SOFTWARE ON A DIFFERENT COMPUTER, YOU MUST CON TACT PCA TO OBTAIN THE NECESSARY REINSTALLATION PROCEDURES.

2. CHARGES AND PAYMENTS

All payments for the Software and User Documentation shall be made to either PCA or Dealer (if any), as appropriate.

3. TERM AND TERMINATION

(a) This Agreement shall be in effect from the date Customer clicks the accepted icon below or installs, copies or otherwise uses the Software or User Documentation until it is terminated by Customer, by Dealer (if any) on behalf of Customer or PCA or by PCA as set forth herein.

(b) This Agreement may be terminated by PCA without cause upon 30 days' written notice or immediately upon notice to Customer if Customer breaches this Agreement or fails to comply with any of its terms or conditions. This Agreement may be terminated by Customer without cause at any time upon written notice to PCA.

4. BACKUP AND REPLACEMENT COPIES

Customer may make one copy of the Software for back-up and archival purposes only, provided PCA's copyright and other proprietary rights notices are included on such copy.

5. PROTECTION AND SECURITY

(a) Customer shall not provide or otherwise make available any of the Software or User Documentation in any form to any person other than employees of Customer with the need to know, without PCA's written permission.

(b) All Software and User Documentation in Customer's possession including, without limitation, translations, compilations, back-up, and partial copies is the property of PCA. Upon termination of this Agreement for any reason, Customer shall immediately destroy all Software and User Documentation, including all media, and destroy any Software that has been copied onto other magnetic storage devices. Upon PCA’s request, Customer shall certify its compliance in writing with the foregoing to PCA.

(c) Customer shall take appropriate action, by instruction, agreement or otherwise, with any persons permitted access to the Software or User Documentation, to enable Customer to satisfy its obligations under this Agreement with respect to use, copying, protection, and security of the same.

(d) If PCA prevails in an action against Customer for breach of the provisions of this Section 5, Customer shall pay the reasonable attorneys' fees, costs, and expenses incurred by PCA in connection with such action in addition to any award of damages.

6. CUSTOMER'S RESPONSIBILITIES

The essential purpose of this Agreement is to provide Customer with limited use rights to the Software and User Documentation. Customer accepts full responsibility for: (a) selection of the Software and User Documentation to satisfy Customer's business needs and achieve Customer's intended results; (b) the use, set-up and installation of the Software and User Documentation; (c) all results obtained from use of the Software and User Documentation; and (d) the selection, use of, and results obtained from any other software, programming equipment or services used with the Software or User Documentation.

7. LIMITED WARRANTIES

PCA and Dealer (if any) warrants to Customer that: (a) PCA and Dealer (if any) has title to the Software and User Documentation and/or the right to grant Customer the rights granted hereunder; (b) the Software and User Documentation provided hereunder is PCA's most current production version thereof; and (c) the copy of the Software provided hereunder is an accurate reproduction of the original from which it was made.

8. INFRINGEMENT

(a) PCA shall defend Customer against a claim that the Software or User Documentation furnished and used within the scope of the license granted hereunder infringes a U.S. patent or U.S. registered copyright of any third party that was issued or registered, as applicable, as of the date Customer clicked the accepted icon below or installed, copied or otherwise began using the Software or User Documentation, and PCA shall pay resulting costs, damages, and attorneys' fees finally awarded, subject to the limitation of liability set forth in Section 9 entitled "Limitation of Remedy," provided that:

1. Customer promptly notifies PCA in writing of the claim.

2. PCA has sole control of the defense and all related settlement negotiations.

3. If such claim has occurred or in PCA's opinion is likely to occur, Customer shall permit PCA at its sole option and expense either to procure for Customer the right to continue using the Software or User Documentation or to replace or modify the same so that it becomes noninfringing. If neither of the foregoing alternatives is reasonably available in PCA's sole judgment, Customer shall, on one month's written notice from PCA, return to PCA the Software and User Documentation and all copies thereof.

(b) PCA shall have no obligation to defend Customer or to pay costs, damages or attorneys' fees for any claim based upon (1) use of other than a current unaltered release of the Software or User Documentation, or (2) the combination, operation or use of any Software or User Documentation furnished hereunder with any other software, documentation or data if such infringement would have been avoided but for the combination, operation or use of the Software or User Documentation with other software, documentation or data.

(c) The foregoing states the entire obligation of PCA and Customer's sole remedy with respect to infringement matters relating to the Software and User Documentation.

9. LIMITATION OF REMEDY

(a) PCA'S AND DEALER'S (IF ANY) ENTIRE LIABILITY AND CUS TOMER'S EX CLUSIVE REMEDY FOR DAMAGES DUE TO PERFORMANCE OR NONPER FORMANCE OF ANY SOFTWARE OR USER DOCUMENTATION, PCA, DEALER (IF ANY), OR ANY OTHER CAUSE WHATSOEVER, AND REGARDLESS OF THE FORM OF ACTION, WHETHER IN CONTRACT OR IN TORT, INCLUDING NEGLIGENCE, SHALL BE LIMITED TO THE AMOUNT PAID TO PCA OR DEALER (IF ANY) FOR THE SOFTWARE AND USER DOCUMENTATION.

(b) NEITHER PCA NOR DEALER (IF ANY) IS AN INSURER WITH RE GARD TO THE PERFORMANCE OF THE SOFTWARE OR USER DOCUMENTA TION. THE TERMS OF THIS AGREEMENT, INCLUDING, BUT NOT LIMITED TO, THE LIMITED WARRANTIES, AND THE LIMITATION OF LIABILITY AND REMEDY, ARE A REFLEC TION OF THE RISKS ASSUMED BY THE PARTIES. IN ORDER TO OBTAIN THE SOFTWARE AND USER DOCUMENTATION FROM PCA OR DEALER (IF ANY), CUSTOMER HEREBY ASSUMES THE RISKS FOR (1) ALL LIABILITIES DISCLAIMED BY PCA AND DEALER (IF ANY) ON THE FACE HEREOF; AND (2) ALL ACTUAL OR ALLEGED DAMAGES IN EXCESS OF THE AMOUNT OF THE LIMITED REMEDY PROVIDED HEREUNDER. THE ESSENTIAL PURPOSE OF THE LIMITED REMEDY PROVIDED CUSTOMER HEREUNDER IS TO ALLOCATE THE RISKS AS PROVIDED ABOVE.

10. U.S. GOVERNMENT RESTRICTED RIGHTS

This commercial computer software and commercial computer software documentation were developed exclusively at private expense by Portland Cement Association, 5420 Old Orchard Road, Skokie, Illinois 60077. U.S. Government rights to use, modify, release, reproduce, perform, display or disclose this computer software and computer software documentation are subject to the restrictions of DFARS 227.7202-1(a) (June 1995) and DFARS 227.7202-3(a) (June 1995), or the Restricted Rights provisions of FAR 52.227-14 (June 1987) and FAR 52.227-19 (June 1987), as applicable.

11. GENERAL

(a) No action arising out of any claimed breach of this Agreement or transactions un der this Agreement may be brought by Customer more than two years after the cause of such action has arisen.

(b) Customer may not assign, sell, sublicense or otherwise transfer this Agreement, the license granted herein or the Software or User Documentation by operation of law or otherwise without the prior written consent of PCA. Any attempt to do any of the foregoing without PCA's consent is void.

(c) Customer acknowledges that the Software, User Documentation and other pro prietary information and materials of PCA are unique and that, if Customer breaches this Agreement, PCA may not have an adequate remedy at law and PCA may enforce its rights hereunder by an action for damages and/or injunctive or other equitable relief without the necessity of proving actual damage or posting a bond therefor.

(D) THE RIGHTS AND OBLIGATIONS UNDER THIS AGREEMENT SHALL NOT BE GOVERNED BY THE UNITED NATIONS CONVENTION ON CONTRACTS FOR THE INTERNATIONAL SALE OF GOODS, THE APPLICATION OF WHICH IS EX PRESSLY EXCLUDED, BUT SUCH RIGHTS AND OBLIGATIONS SHALL INSTEAD BE GOVERNED BY THE LAWS OF THE STATE OF ILLINOIS, APPLICABLE TO CON TRACTS ENTERED INTO AND PERFORMED ENTIRELY WITHIN THE STATE OF ILLINOIS AND APPLICABLE FEDERAL (U.S.) LAWS. UCITA SHALL NOT APPLY TO THIS AGREEMENT.

(E) THIS AGREEMENT SHALL BE TREATED AS THOUGH IT WERE EXE CUTED IN THE COUNTY OF COOK, STATE OF ILLINOIS, AND WAS TO HAVE BEEN PERFORMED IN THE COUNTY OF COOK, STATE OF ILLINOIS. ANY ACTION RELATING TO THIS AGREEMENT SHALL BE INSTITUTED AND PROSECUTED IN A COURT LOCATED IN COOK COUNTY, ILLINOIS. CUSTOMER SPECIFICALLY CONSENTS TO EXTRATERRITORIAL SERVICE OF PROCESS.

(f) Except as prohibited elsewhere in this Agreement, this Agreement shall be binding upon and inure to the benefit of the personal and legal representatives, permitted successors, and permitted assigns of the parties hereto.

(g) All notices, demands, consents or requests that may be or are required to be given by any party to another party shall be in writing. All notices, demands, consents or requests given by the parties hereto shall be sent either by U.S. certified mail, postage prepaid or by an overnight international delivery service, addressed to the respective parties. Notices, demands, consents or requests served as set forth herein shall be deemed sufficiently served or given at the time of receipt thereof.

(h) The various rights, options, elections, powers, and remedies of a party or parties to this Agreement shall be construed as cumulative and no one of them exclusive of any others or of any other legal or equitable remedy that said party or parties might otherwise have in the event of breach or default in the terms hereof. The exercise of one right or remedy by a party or parties shall not in any way impair its rights to any other right or remedy until all obligations imposed on a party or parties have been fully performed.

(i) No waiver by Customer, PCA or Dealer (if any) of any breach, provision, or de fault by the other shall be deemed a waiver of any other breach, provision or default.

(j) The parties hereto, and each of them, agree that the terms of this Agreement shall be given a neutral interpretation and any ambiguity or uncertainty herein should not be construed against any party hereto.

(k) If any provision of this Agreement or portion thereof is held to be unenforceable or invalid by any court or competent jurisdiction, such decision shall not have the effect of invalidating or voiding the remainder of this Agreement, it being the intent and agreement of the parties that this Agreement shall be deemed amended by modifying such provision to the extent necessary to render it enforceable and valid while preserving its intent or, if such modification is not possible, by substituting therefor another provision that is enforceable and valid so as to materially effectuate the parties' intent.

(l) Except as set forth herein, this Agreement may be modified or amended only by a written instrument signed by a duly authorized representative of PCA and Customer.

PcaStructurePoint Contact Information

Web Site:	http://www.pcaStructurePoint.com
E-mail:	info@pcaStructurePoint.com support@pcaStructurePoint.com
Ordering:	pcaStructurePoint Attn: Customer Service 5420 Old Orchard Road Skokie, IL 60077 USA
Phone:	(847) 972-9042
Fax:	(847) 972-9043
Technical Support:	pcaStructurePoint 5420 Old Orchard Road Skokie, IL 60077 USA
Phone:	(847) 966-HELP / (847) 966-4357
Fax:	(847) 581-0644

Bug Report Form

BUG REPORT FORM

Program Name:		Company Name:	
Program Version:		Tester's Name:	
Program Release:		Tel:	
Operating System:		Fax:	
CPU / Memory:		E-mail:	
Network:			

Bug No.		Input File:	
Date:		Output File:	

Bug Priority Category:		Screen Shot File:	

Problem Title: Please enter a brief one-line description of the problem.

Summary Information: Restate the problem title and/or include more descriptive summary information.

Error Messages: If the problem causes any error messages, please list the exact error messages that you are receiving.

Steps to Reproduce: If the problem is reproducible, please list the steps required to cause it. If the problem is not reproducible (only happened once, or occasionally for no apparent reason), please describe the circumstances in which it occurred and the symptoms observed.

Results: Describe your results and how they differed from what you expected.

Workaround: If there is a workaround for the problem, please describe it in detail.

Documentation & Notes: Document any additional information that might be useful in resolving the problem.

Introduction to pcaSlab

Program Features

pcaSlab is a computer program for the analysis and design of reinforced concrete slab systems. Using the Equivalent Frame Method, an equivalent two-dimensional frame of up to 20 spans can be analyzed and designed. In addition to the design option **pcaSlab** has the capability of investigating existing slab systems. **pcaSlab** analyzes the input floor system and outputs the shear and moment at all critical sections. Punching shear is also checked, and cracked-section or gross-section deflections are calculated. Material quantity take-offs are computed. In addition to the required area of reinforcing steel at the critical sections, **pcaSlab** provides a complete bar schedule that includes number of bars and bar sizes and lengths. **pcaSlab** checks all applicable provisions of the relevant code.

General Features

- Online help
- Checking of data as they are input
- Graphical display of geometry and loads as they are input
- Auto-input feature that walks you through the input process
- User-controlled screen color settings
- Ability to save defaults and settings for future input sessions
- Print preview of graphical screen
- Mixed slab system types
- Customizable results report
- Import input data from ADOSS

Program Capacity

Input

Material properties

Concrete (for slabs, beams, and columns):

- Unit weight, w
- Compressive strength, f'_c

- Modulus of elasticity, E_c
- Modulus of rupture, f_r

Steel:

- Longitudinal reinforcement yield strength, f_y
- Transverse reinforcement yield strength, f_{yv}
- Plain or epoxy coated bars

Structure geometry

- Up to 20 spans
- May include two cantilevers
- Slab thickness may vary from span to span
- Transverse design strip width (left and right) may vary from span to span
- Round or rectangular columns above and below the slab
- Column height may vary
- User-controlled column-fixity
- Standard or user-input drop panels
- Column capitals
- Longitudinal beams
- Transverse beams
- Waffle slab rib width, depth, and clear spacing

Loads

- Self weight automatically computed
- Uniform dead and live surface (force per unit area) loads
- Partial loads may be uniform (line), trapezoidal, concentrated, or moments
- Lateral loads effect may be input as joint moments
- Pattern loading—live load percentage may be modified
- Up to six load cases
- Up to twelve load combinations

Design parameters

The following are specified for slab and beam top and bottom bars:

- Bar cover
- Minimum bar size
- Maximum bar size
- Minimum and maximum clear spacing between bars
- Minimum and maximum reinforcement ratios

Output

- Ability to view results prior to printing
- Ability to print graphical views (plan, elevation, side view, and isometric frame views, and shear, moment, and deflection diagrams)
- User customized result report may include some or all of the following:
    - Echo of input data
    - Design results
    - Column service load (moments and axial loads)
    - Segmental shear and moment, for each load combination
    - Segmental moment and shear envelope
    - Segmental deflection

Slab system may contain any combination of

- Drop Panels
- Column Capitals
- Transverse Beams
- Longitudinal Beams
- Ribs (Waffle Slabs)

System of units

- US (in.-lb)
- SI (metric)

Code

- ACI 318-99; ACI 318M-99; ACI 318-02; ACI 318M-02
- CSA A23.3-94; CSA A23.3-94E

Graphical Views

- Plan, elevation, side view and isometric views of the modeled frame, with ability to zoom (enlarge)
- Graphical representation of dead and live loads
- Shear and moment diagrams
- Shear and moment envelopes (minima and maxima)
- Shear and moment capacity
- Reinforcement
- Deflection

System Requirements

Operating systems

- Microsoft Windows 95
- Windows 98
- Windows ME
- Windows NT4
- Windows 2000
- Windows XP

Minimum Requirements

- 100 MHz processor
- 32 MB of RAM memory
- 50 MB of free hard disk space for program installation
- A monitor supported by one of the operating systems listed above
- A mouse or other pointing device supported by one of the operating systems listed above

Recommended Options

- A printer supported by one of the operating systems listed above with the capability of printing bitmaps
- Most recent OS service pack installed

Terms and Conventions

Terms

The following terms are used throughout this manual. A brief explanation is given to help familiarize yourself with them.

Term	Meaning
Windows	Refers to the Microsoft Windows 95 or higher.
[ ]	Indicates equivalent Canadian Code requirements.
Click on	Means to position the mouse cursor on top of a designated item or location and to press and release the left-mouse button.
Right-click on	Means to position the mouse cursor on top of a designated item or location and to press and release the right-mouse button.
Double-click on	Means to position the cursor on top of a designated item or location and to press and release the left-mouse button twice in quick succession.
Marquee select	Means to depress the mouse button and continue to hold it down while moving the mouse. As you drag the mouse, a rectangle (known as a marquee) follows the cursor. Release the mouse button and the area inside the marquee is selected.

Conventions

Various styles of text and layout have been used in this manual to help differentiate between different kinds of information. The styles and layout are explained below.

Style	Meaning
ALL CAPS	Indicates the name of a path, a directory, or a file. For example, C:\PCAMATS\PCAMAT1.EXE refers to the file "PCAMAT1.EXE" that exists on your "C" drive in the "PCAMATS" folder. Windows environments are NOT case sensitive. Thus, "pcamat1.exe" and "PCAMAT1.EXE" refer to the same file.
Bold	Bold typeface makes reference to either a menu or a menu item command such as File or Save, or a button such as Ok and Cancel, or a key combination such as ALT+F.
Italic	Indicates a subtitle in a chapter, a glossary item, or emphasizes. The chapter subtitles are italic and bold.
KEY + KEY	Indicates a key combination on your keyboard. The plus sign indicates that you should press and hold the first key while quickly pressing the second key and then release both. For example "ALT + F" indicates that you should press the "ALT" key on your keyboard and hold it while you press the "F" key, then release both keys.
Mono-space	Indicates something you should enter with the keyboard. For example, type "c:*.txt".

Ch. 1 Installing the Program

Running Program Installation

Installation Procedures:

1. If you have the installation disk please insert it into the CD drive and proceed to step 2. If you do not have the installation disk but downloaded the installation file from our website, please run the file and proceed to step 11.
2. If the auto-run functionality of Windows is enabled on your computer (which is the default setting) and a web browser (Internet Explorer, Netscape, etc.) is installed, the start page (Figure 1-1) is shown automatically. If the start page does not show up automatically, please go to step 7.

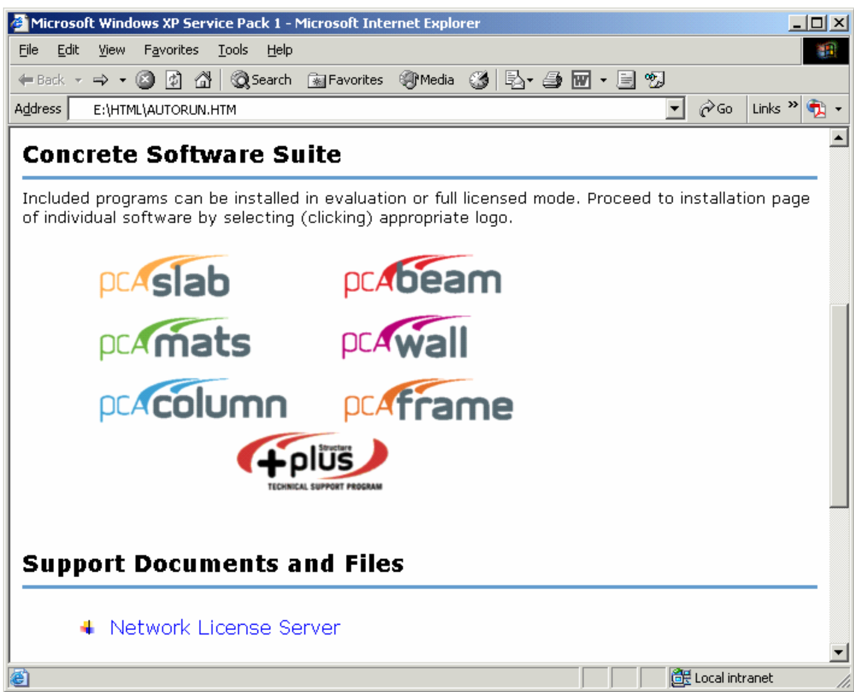


Figure 1-1 Start Page

3. If you are going to evaluate PCA software, please click the **Evaluation Software License Agreement** link and read through the agreement carefully. If you are going to use a commercial copy of PCA software, please click the **Software License Agreement** link and read through the agreement carefully.

4. Click pcaslab to start the installation. A second page will be shown as Figure 1-2.

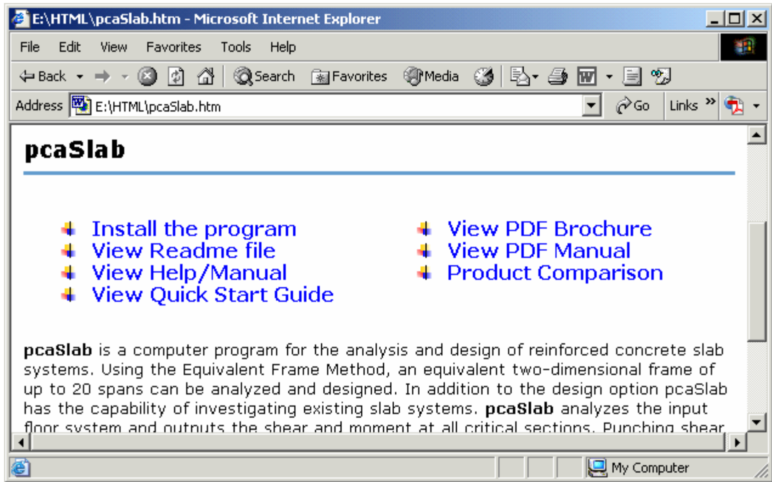


Figure 1-2 Second Installation Page

5. Click the **Install the program**, then a dialog box will be shown as Figure 1-3.

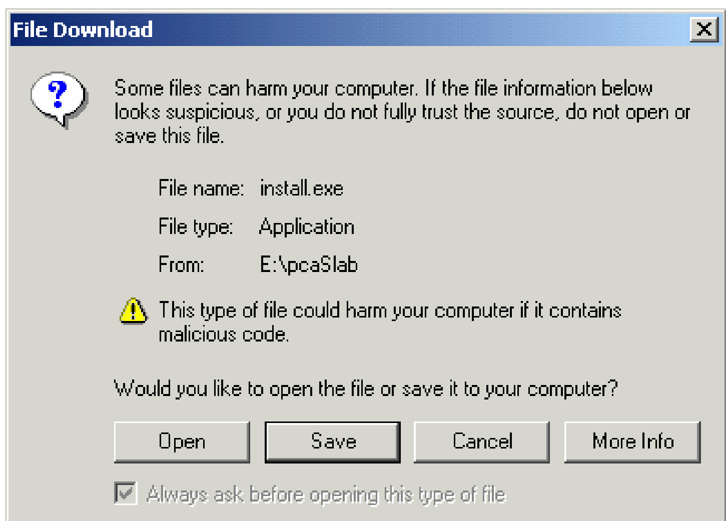


Figure 1-3 Click Open to Continue

6. Click the  button on Figure 1-3 to continue. You may ignore the following steps and directly go to step 11 below.
7. If the start page doesn't show up automatically, open the Windows Explorer.

   Windows 95, 98, ME, NT, 2000: **Start/Programs/Accessories/Windows Explorer**

   Windows XP: **Start/All Programs/Accessories/Windows Explorer**

   You may also start the Windows Explorer by pressing the keyboard short cut of  **+ E.** A window similar to Figure 1-4 appears.

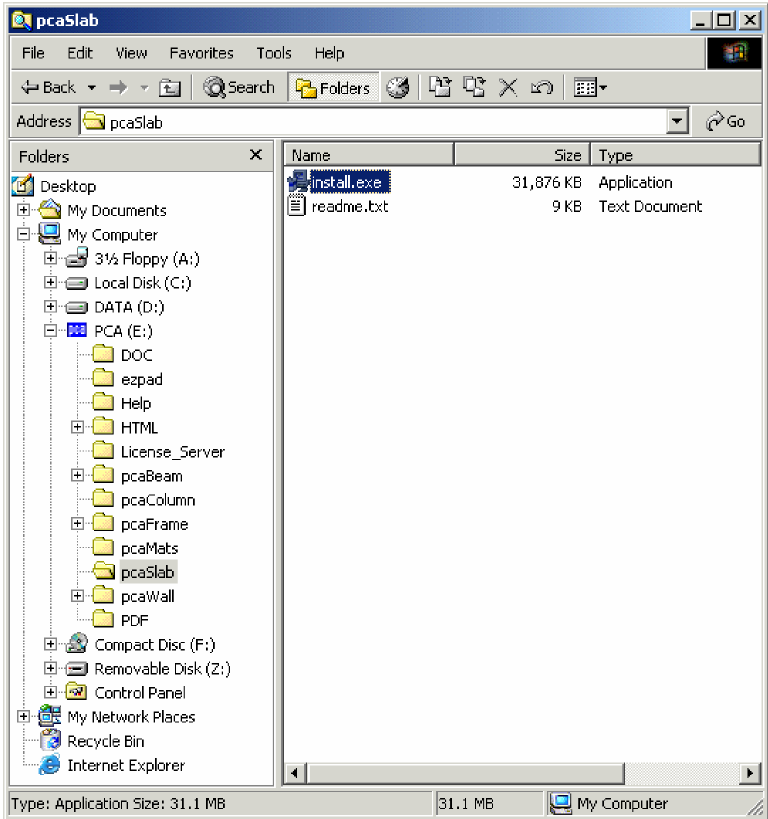


Figure 1-4 Windows Explorer

8. Select the CD drive from the left pane of Windows Explorer (e.g. E drive in Figure 1-4).
9. Select the **pcaSlab** folder from the left pane. The contents of this folder are shown in the right pane.
10. Double click the **install.exe** in the right pane of the Windows Explorer.
11. The installation process starts by displaying the following window (Figure 1-5). Please read all the information. Then press the Next > button. This will continue the installation process by installing license manager, an evaluation license, and the application software.

Figure 1-5 Beginning of the installation

12. The first step of the application installation process is to review the information on the software and the copyrights announcement as shown in Figure 1-6. Please read all the information carefully. Press the I Agree button to confirm that you have read and agreed with it. This will continue the installation. If you do not agree with the license agreement, press the I Do Not Agree button. This will stop the installation.

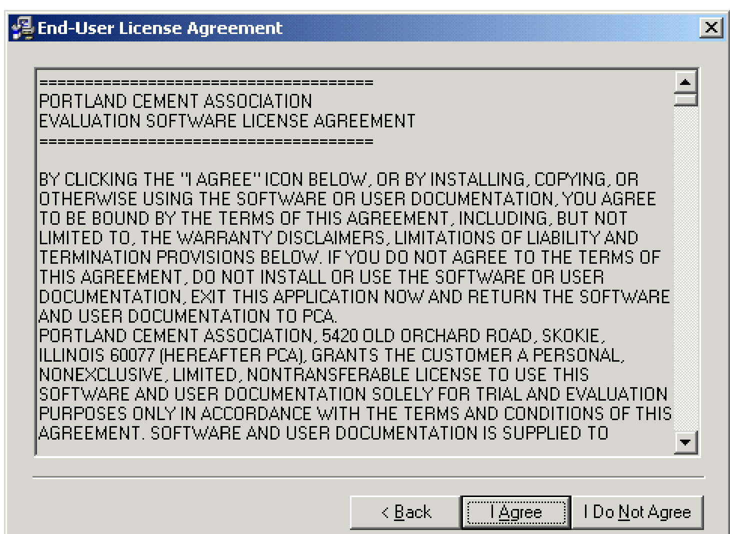


Figure 1-6 Copyright announcement

13. Review the Readme file (Figure 1-7), and then press the Next > button to continue.

Figure 1-7 Readme File

14. Enter the registration information (Figure 1-8). This information will be shown in the **Help | About** dialog box. Press the Next > button to go to the next step.

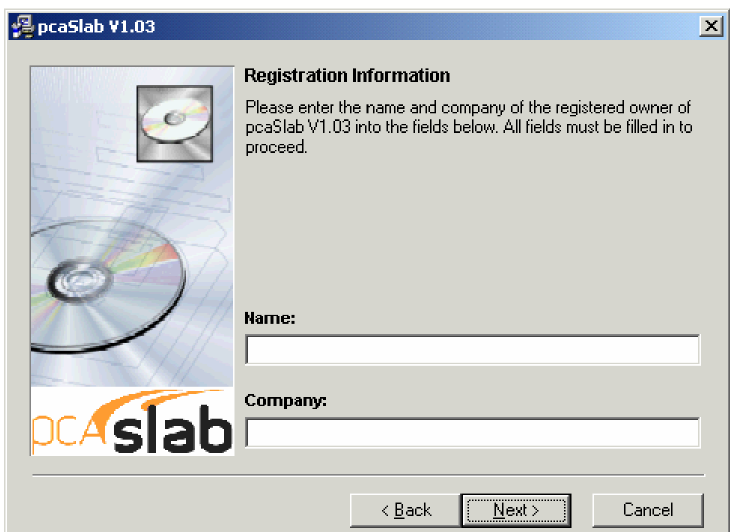


Figure 1-8 Enter Registration Information

15. If you continue the installation then the next setup is to decide the directory where pcaSlab is to be installed as shown in Figure 1-9. The default directory is **C:\Program Files\PCA\pcaSlab**. You may press the Browse... button to locate the directory. If the directory does not exist, the setup program will create it. Press the Next > button to go to the next step.

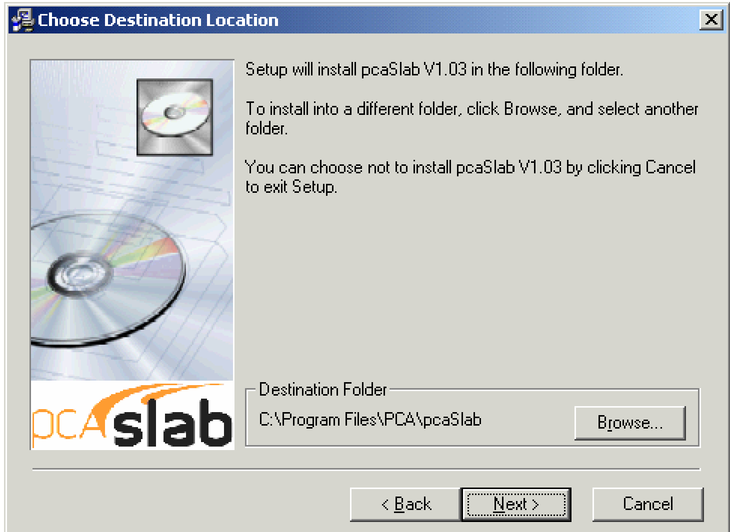
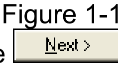


Figure 1-9 Choose Destination Location

16. The next step is to enter the group name as shown in Figure 1-10.Windows will use this name in the **Start/Programs** menu. Press the  button to go to the next step.

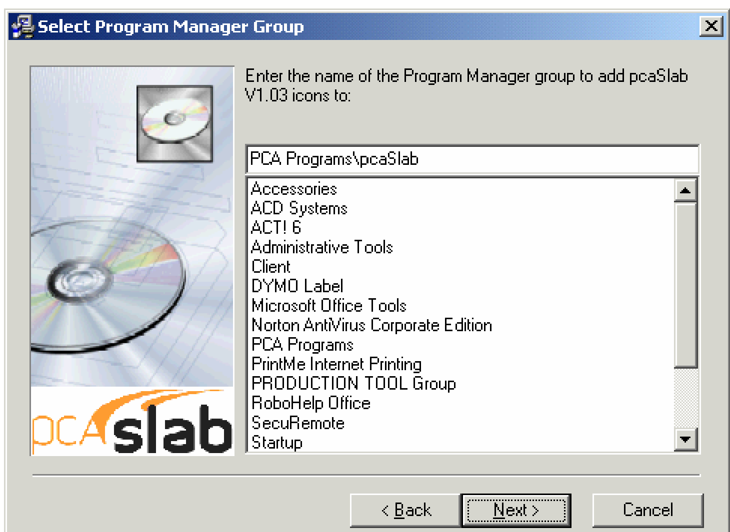


Figure 1-10 Select Program Manager Group

17. After all the previous steps are completed, press the  button as shown in Figure 1-11 to start the installation.

Figure 1-11 Start Installation

18. During the installation a window as in Figure 1-12 shows the progress as files are copied to your hard drive.

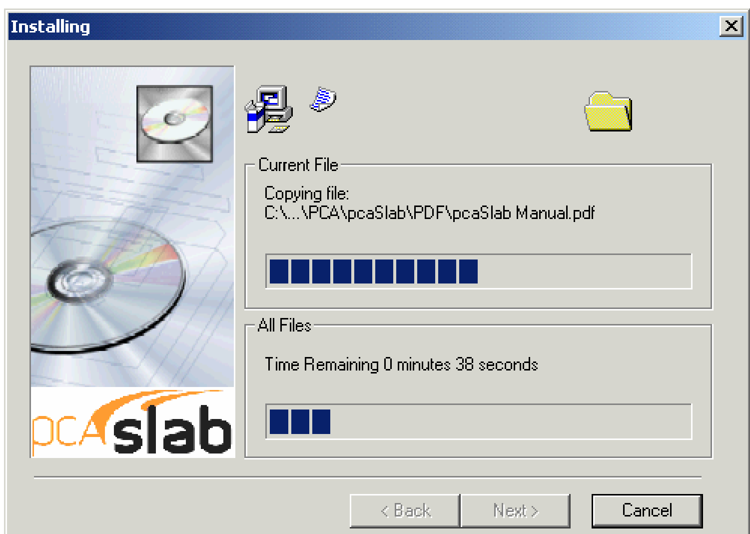


Figure 1-12 Installation progress

19. After the installation is completed, a dialog box similar to Figure 1-13 is shown. Press the Finish > button to finish the installation.

Figure 1-13 Installation Complete

Purchasing and Licensing Process

The following information is for stand-alone license only. Please refer to the License server Quick Start Guide in the License Server directory on CD regarding the procedures to install networking license.

Licensing Model

By default, each pcaStructurePoint software application comes in an Evaluation mode. This means that initially our products can be used for a limited time of 15 days. If the user decides to purchase the software license, and completes the purchase, pcaStructurePoint will provide a License Code. By entering this code, user will activate the software license. Once the software license is activated the application will no longer have restrictions limiting the time of operation.

Evaluation mode

This is the default mode of the license. In this licensing scheme, users are granted limited rights to use our full-featured software. The only visible difference between the evaluation version and the licensed version is an additional start-up dialog box as shown in Figure 1-14. It gives a choice of buying the software, entering the activation code or running the application in the evaluation mode by pressing the Evaluate button. Evaluation mode is available for a limited time only. By default, pcaStructurePoint software applications come with 15 days. That means, from the day of installation, a user can evaluate the software for the next 15 days. After this time, user will either obtain a License, or uninstall the application.

Note: Any tampering with system clock or evaluation license file will render the software useless.

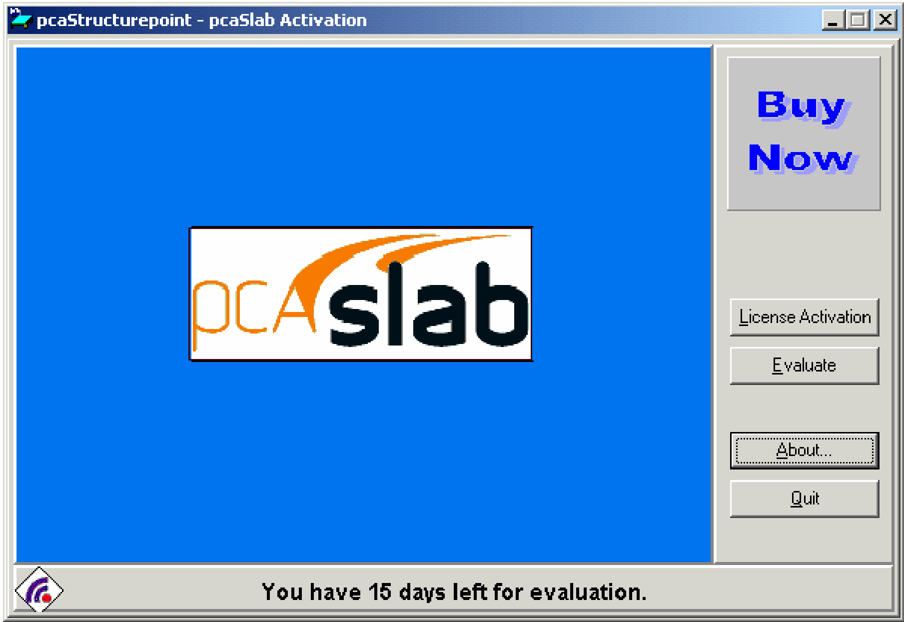


Figure 1-14 Start-up Dialog Box

How to Purchase

You may purchase the license on-line on our web site at www.pcaStructurePoint.com or by calling our Sales Department at 847-972-9024. To buy on-line you may press **BUY NOW** button which will run your default web browser and open the product web page where you will be able to complete the transaction.

Licensing Activation

After the purchase is completed, pcaStructurePoint will generate a unique Activation Code based on the product ID and the locking code, which is derived from the unique hardware-id of the user's computer. Each License is associated with the particular PC in which user has installed the pcaStructurePoint software. This implies that the activation-code will not work on any other PC. You can transfer the license to another PC by contacting pcaStructurePoint for the transfer procedures.

To activate the license press the License Activation button in the start-up dialog box. This will bring a window as in Figure 1-15, which shows three license activation methods.

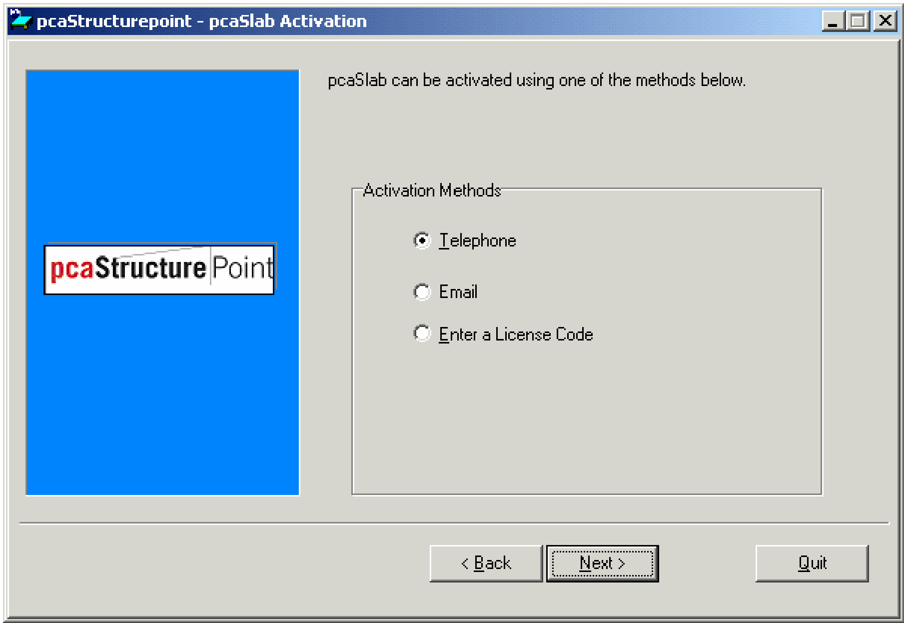


Figure 1-15 License activation methods

If you have already received the license code you may choose **ENTER LICENCE CODE** and then press the Next > button. A window (Figure 1-16) will pop up where you will be able to type in the license code. However, in order to avoid mistyping we advise to use copy and paste feature instead of typing the code.

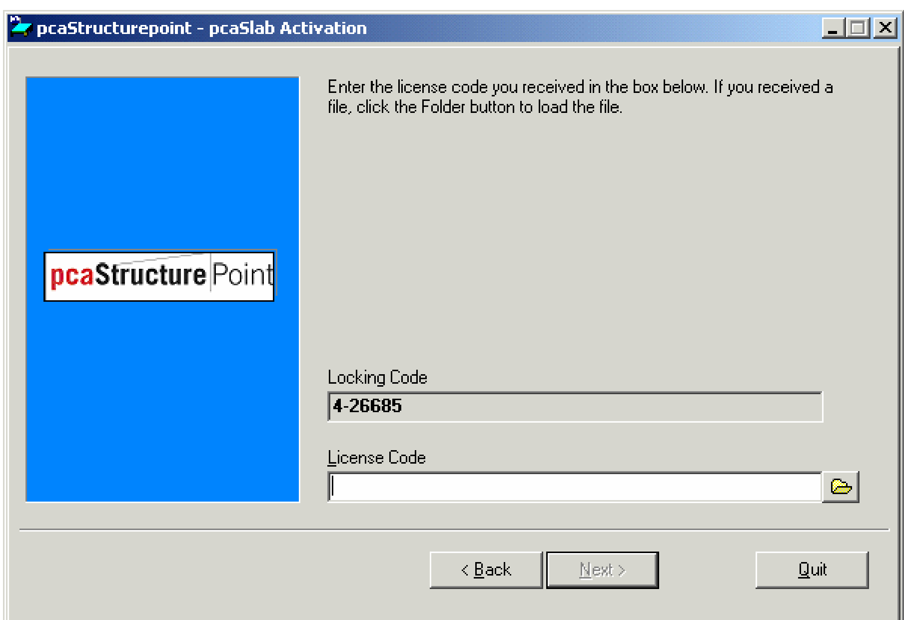


Figure 1-16 Enter license code

The license code can also be extracted from a file. To do that press the button and open the file in which the license code is stored using the open window (Figure 1-17).

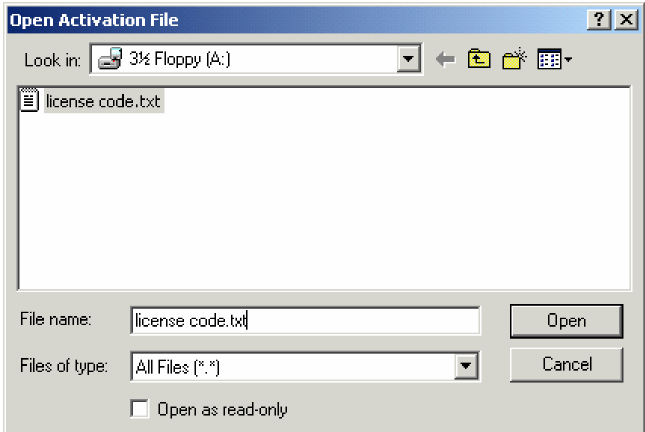
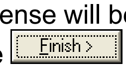


Figure 1-17 Open activation file

If the entered code is correct the license will be activated and window as in Figure 1-18 will show. When you press the 

button the license activation will be completed.

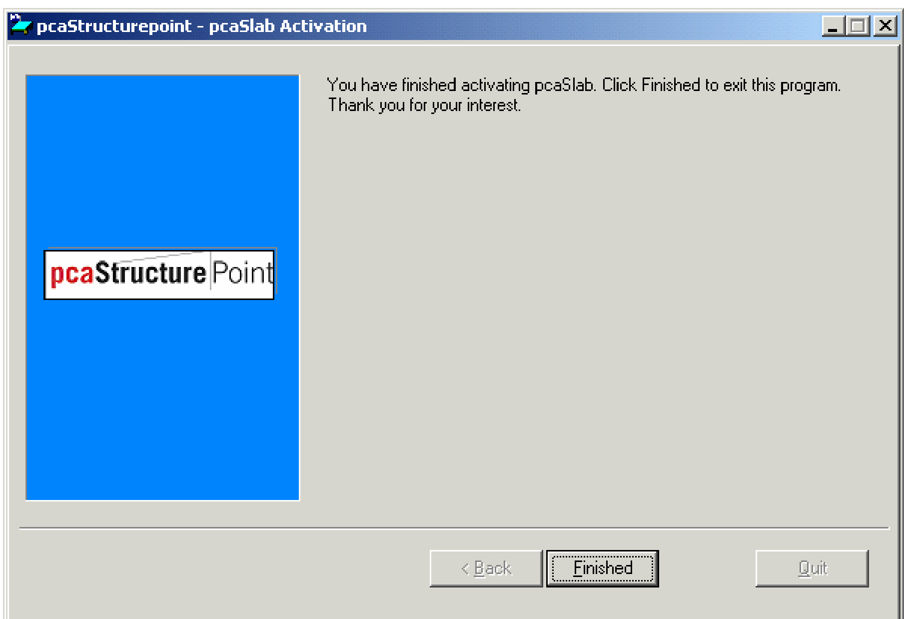


Figure 1-18 Activation completion

If you do not have the license code you may chose either **TELEPHONE** or **E-MAIL** method (Figure 1-15) whichever is more convenient for you. Please note that for **E-MAIL** method you need to have the Internet connection and a default mailer configured in your system.

Activation by Phone

If the telephone method was chosen to activate the license the following screen appears (Figure 1-19). It shows the product ID and the Locking Code. This information is needed when you make a phone call at 847-966-4357 (HELP) to obtain the license. You will be asked to provide information about yourself in order to verify that you have purchased the license.

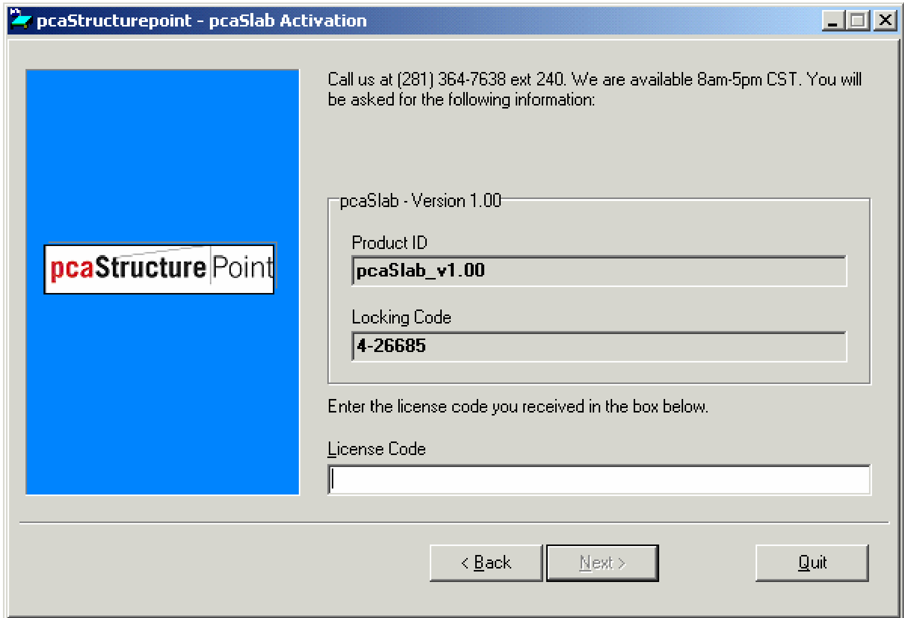


Figure 1-19 Activation by phone

When your information is verified, the license code will be generated and provided to you by an email. When you receive the code type it or copy-paste it in the **LICENCE CODE** edit box and press the Next > button. If the entered code is correct, the license will be activated and window as in Figure 1-18 will show. When you press the Finish > button the license activation will be completed.

Activation by e-mail

If the e-mail method was chosen to activate the license a screen (Figure 1-20) will show prompting you to provide information about yourself. When you type in all the information press the Send E-mail button and it will be automatically e-mailed to us together with the product ID and the Locking Code. After the information is verified the license code will be generated and e-mailed back to you.

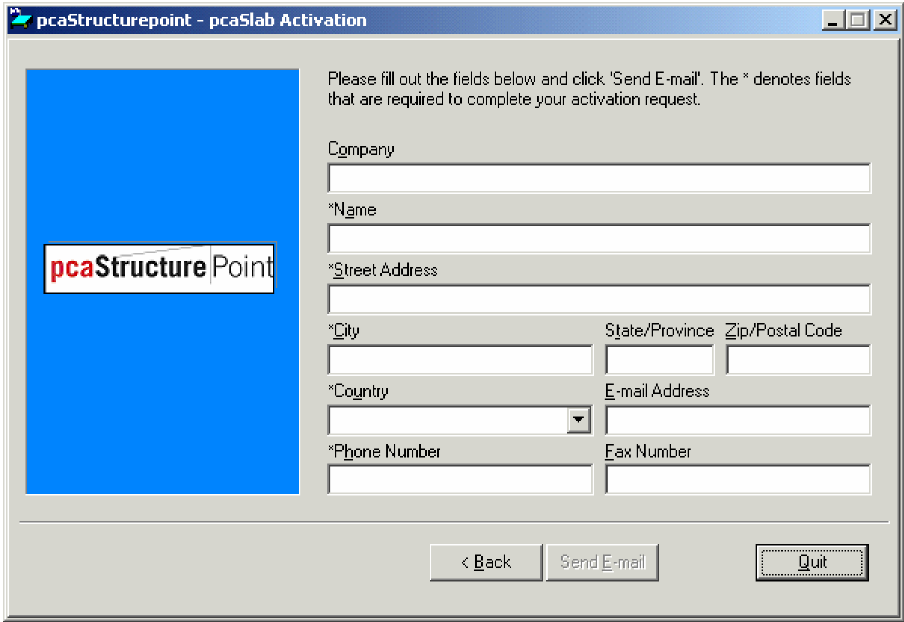


Figure 1-20 Activation by e-mail

After you receive it you may enter it by pressing the License Activation button in the start up dialog box (Figure 1-14) and choosing **ENTER LICENCE CODE** option. If the entered code is correct, the license will be activated and window as in Figure 1-18 will show. When you press the Finish > button the license activation will be completed.

If you experience any problems registering or licensing the software, contact our Technical Support team.

Running the Program

To execute pcaSlab from within Windows:

1. Windows 95, 98, NT, 2000:

   Select the  button from the lower left corner of the screen then follow the path of **Programs/PCA Programs/pcaSlab/**.
2. Windows XP:

   Select the  button from the lower left corner of the screen then follow the path of **All Programs/PCA Programs/pcaSlab/**.
3. Select the "**pcaSlab**" icon.

Removing the Program

To remove the program from your computer:

1. Click  button (Windows 95, 98, NT, 2000) or  button (Windows XP) on the lower left corner of the screen.
2. In Windows 95, 98, NT, 2000, select **Programs/PCA Programs/pcaSlab/**; In Windows XP, select **All Programs/PCA Programs/pcaSlab/**.
3. Select **Uninstall pcaSlab**.
4. Follow the steps shown by the un-installation wizard to remove the program.

Ch. 2 Method of Solution

Method of Solution

The user should be aware of the assumptions made by the program during the design stage. These include details regarding loading, strip widths, reinforcement selection, deflection computations, material quantities, etc.

Geometric Consistency Checks

The **pcaSlab** program provides geometric checks to avoid an analysis with an inconsistent system. The dimensions of the slabs, drops and column capitals are checked and modified to produce a legitimate system.

Geometric Code Checks

Prior to an analysis, **pcaSlab** checks that the input values for the drop panel dimensions, column capital dimensions, T-beam flange widths, waffle slab rib dimensions and slab and beam depths meet the applicable code values.

Slab Dimensions

If the slab cantilever length is less than one-half the column dimension in the direction of analysis (c_1), or less than the lateral extension of the transverse beam into the cantilever, the cantilever length will be increased to the larger of these two lengths. If the slab width is less than one-half the column dimension transverse to the direction of analysis (c_2), or less than one-half the longitudinal beam width, the slab width will be increased to the larger of these two widths.

Drop Panel Dimensions

If the drop panel lengths extend beyond the end of the slab cantilevers, the drop panel lengths will be reduced so that they extend only to the cantilever tip. The drop panel edges will be shifted forward or backward in the transverse direction when the slab strip width on either side of the column is less than one-half the drop panel width.

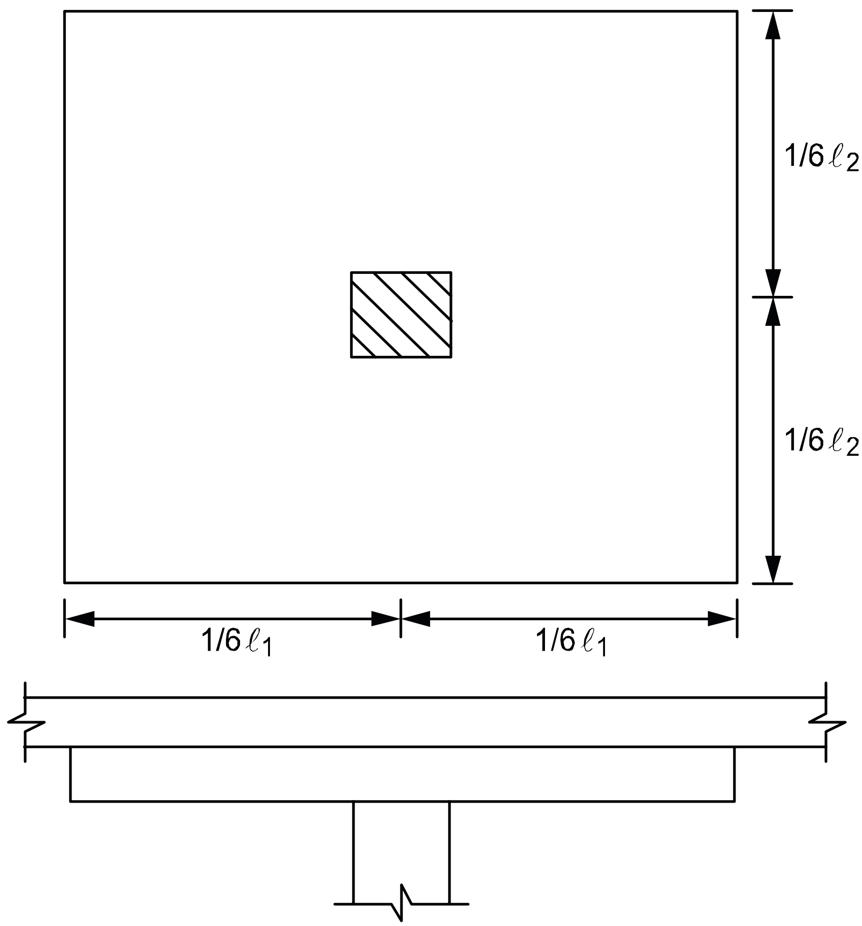


Figure 2-1 Valid drop dimensions

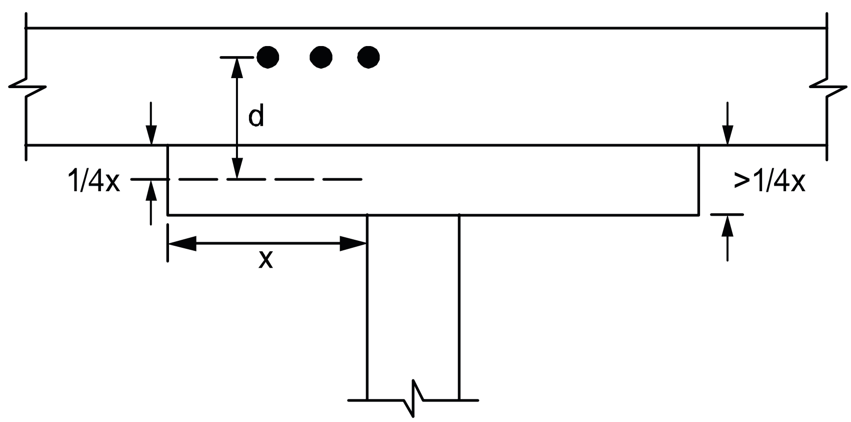


Figure 2-2 Excess drop depth

A valid drop must extend in each direction at least one-sixth the center-to-center span length in that direction (Figure 2-1) and have a drop depth below the slab of at least one-quarter the input slab depth. ACI slabs that contain valid drops are allowed a 10%

decrease in minimum slab depth[1]. The drop depth of an invalid drop will not be used in the calculation of the depth used to reduce the amount of reinforcement required at a column[2], and will not be used in the calculation of moment of inertia for deflection computations.

If the valid drop depth is greater than one-quarter the distance from the edge of the drop panel to the face of the column (x) the excess depth exceeding ¼x will not be considered in the calculation of the effective depth used to reduce the amount of reinforcement required at a column (Figure 2-2).[3]

The input drop dimensions will be used for self-weight computations, when computing slab stiffness and to determine moments and shears when computing punching shear around a column.

Column Capital Dimensions

If a column capital contains a depth/extension ratio less than 1, the extension will be limited to the capital depth. The upper limit for the depth/extension ratio is 50. If, by drawing lines extending the column capital through the drop or beam to the underside of the slab, the column capital extension falls outside the edge of the drop or beam at the slab soffit, the extension will be modified such that these lines extend only up to the edge of the drop or beam at the slab soffit (Figure 2-3). The modified column capital dimensions will be used when computing column stiffness.

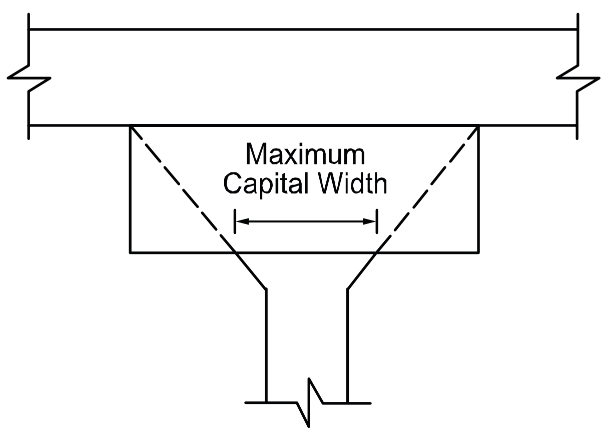


Figure 2-3 Maximum capital width

[1] ACI 318-99, 9.5.3.2; ACI 318-02, 9.5.3.2

[2] ACI 318-99, 13.3.7; ACI 318-02, 13.3.7

[3] ACI 318-99, 13.3.7.3; ACI 318-02, 13.3.7.3; CSA A23.3-94, 13.11.6

Minimum Slab Thickness of Flat Plate, Flat Slab and Beam-Supported Slab Systems

For ACI 318 Code, the minimum thickness of slabs with or without interior beams spanning between supports and having a ratio of long to short span not exceeding 2 is:[4]

$$h = \frac{l_n\left(0.8 + \frac{f_y}{200,000}\right)}{36 + 5\beta\left(\alpha_m - 0.2\right)} \qquad 0.2 < \alpha_m \leq 2.0 \qquad \text{Eq. 2-1}$$

But not less than

$$h = \frac{l_n\left(0.8 + \frac{f_y}{200,000}\right)}{36 + 9\beta} \qquad \alpha_m > 2.0 \qquad \text{Eq. 2-2}$$

For CSA A23.3-94

$$h_s \geq \frac{l_n\left(0.6 + \frac{f_y}{1000}\right)}{30 + 4\beta\alpha_m} \qquad \alpha_m \text{taken} \leq 2.0 \qquad \text{Eq. 2-3}$$

Where

l_n = clear span in the direction of analysis

β = ratio of the clear spans in long to short direction

f_y = yield stress of reinforcing steel

---

[4] ACI 318-99, 9.5.3.3; ACI 318-02, 9.5.3.3; CSA-A23.3-94, 13.3.5

α_m = average value of α, the ratio of flexural stiffness of a beam section to the flexural stiffness of a width of slab bounded laterally by centerlines of adjacent panels on either side of the beam, for all beams supporting the edges of a slab panel.

For the design of ACI slabs without beams ($\alpha_m \leq 0.2$) spanning between supports the minimum thickness shall conform to ACI 318 Table 9.5(c). For flat slabs that contain valid drops, Table 9.5(c) reduces the minimum thickness by approximately 10%.[5] (See Figure 2-1).

The minimum thickness in a span that contains a discontinuous edge will be increased by 10%, if the edge beam provided has a stiffness ratio, α, of less than 0.80.[6] The first and last spans are considered to contain a discontinuous edge as well as a span that contains an exterior edge.

For the CSA Standard the minimum thickness is:

a) For flat plates and slabs with column capitals

$$h_s > \frac{l_n\left(0.6 + \frac{f_y}{1000}\right)}{30} \qquad \text{Eq. 2-4}$$

b) for slabs with drop panels

$$h_s \geq \frac{l_n\left(0.6 + \frac{f_y}{1000}\right)}{30\left[1 + \left(\frac{2x_d}{l_n}\right)\left(\frac{h_d - h_s}{h_s}\right)\right]} \qquad \text{Eq. 2-5}$$

Where $x_d/(l_n/2)$ is the smaller of the values determined in the two directions, x_d shall not be taken greater than $l_n/4$ and h_d - h_s shall not be greater than h_s.

For flat plate systems, the minimum allowable thickness can in no case be less than 5.0 in. For two-way flat slab systems with drops, described above, the minimum allowable

---

5 ACI 318-99, 9.5.3.2; ACI 318-02, 9.5.3.2

6 ACI 318-99, 9.5.3.3 (d); ACI 318-02, 9.5.3.3 (d)

thickness can in no case be less than 4.0 in.[7] For supported slab systems supported by beams with an α_m greater than or equal to 2.0, the minimum allowable thickness can in no case be less than 3.5 in.[8]

A message will be printed with the output showing the required slab depth for any span where the depth does not meet or exceed code requirements.

Minimum Thickness for Waffle Slab Systems

The minimum slab thickness allowed for waffle slabs is one-twelfth the clear rib spacing, or 2 in.[9]

A message will be printed with the output showing the required slab depth for the system if the depth does not meet or exceed code requirements.

Waffle Rib Dimensions

Waffle slab rib dimensions will be considered valid if the rib width is at least 4 in. (or 100 mm), the depth is no more than 3-1/2 times the rib width, and the clear spacing between ribs does not exceed 30 in. (or 750 mm) (Figure 2-4). When valid ribs exist, the ACI code permits the nominal concrete shear strength, V_c, to be increased by 10%.[10]

---

[7] ACI 318-99, 9.5.3.2; ACI 318-02, 9.5.3.2

[8] ACI 318-99, 9.5.3.3; ACI 318-02, 9.5.3.3

[9] ACI 318-99, 8.11.6.1; ACI 318-02, 8.11.6.1

[10] ACI 318-99, 8.11.8; ACI 318-02, 8.11.8;

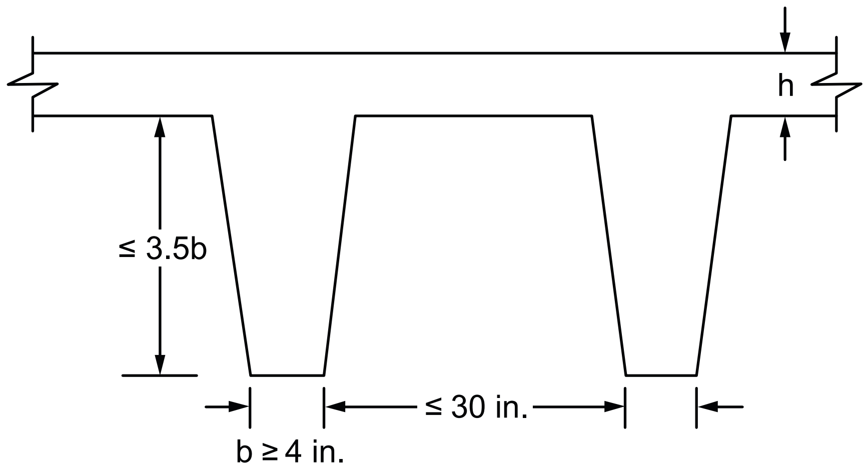


Figure 2-4 Valid rib dimensions

Special Considerations for Waffle Slabs

For the purposes of analysis and design, **pcaSlab** replaces the waffle with solid slabs of equivalent moment of inertia, weight, and punching shear resistance.

The equivalent thickness based on system weight is used to compute the system self-weight. This thickness, h_w, is given by:

$$h_w = \frac{V_{mod}}{A_{mod}} \qquad \text{Eq. 2-6}$$

Where

V_{mod} = the volume of one waffle module.

A_{mod} = the plan area of one waffle module.

The equivalent thickness based on moment of inertia is used to compute slab stiffness. The ribs spanning in the transverse direction are not considered in the stiffness computations. This thickness, h_{MI}, is given by:

$$h_{MI} = \left(\frac{12I_{rib}}{b_{rib}}\right)^{\frac{1}{3}} \qquad \text{Eq. 2-7}$$

Where

I_{rib} = moment of inertia of one waffle section between centerlines of ribs.

b_{rib} = the center-to-center distance of two ribs (clear rib spacing plus rib width).

The drop depth for waffle slab systems is set equal to the rib depth. The equivalent drop depth based on moment of inertia, d_{MI}, is given by:

$$d_{MI} = h_{MI} + h_{rib} \qquad \text{Eq. 2-8}$$

Where

h_{rib} = rib depth below slab.

h_{MI} = equivalent slab thickness based on moment of inertia.

A drop depth entered for a waffle slab system other than 0 will be added to d_{MV}, thus extending below the ribs.

The equivalent thickness based on shear area is used to compute the area of concrete section resisting shear transfer, A_c around the drop. The equivalent slab thickness, h_V, used to compute A_c, is given by:

$$h_v = \frac{A_{rib}}{b_{rib}} + d_{reinf} \qquad \text{Eq. 2-9}$$

Where:

A_{rib} = the entire rib area below the slab plus the slab thickness minus the distance to the reinforcement centroid, d_{reinf}, within the rib width, (i.e., the slab depth between the ribs is not considered as contributing to shear resistance.)

b_{rib} = the center-to-center distance of two ribs (clear rib spacing plus rib width).

d_{reinf} = the distance to reinforcement centriod from the slab top at the support.

Material Properties

By entering the concrete density and compressive strength of the members, default values for the other concrete properties are determined. The slab, column, and beam members may have different concrete properties.

The density of concrete is used to determine the type of concrete, modulus of elasticity, and self-weight.

For the ACI 318 Code, the concrete type is used to determine the default value of f_{ct}, the average split tensile strength of concrete. The concrete type is determined in accordance with Table 2-1.

Note: The CSA Standard does not use f_{ct}

Density (w)		Type
pcf	kg/m3	
w ≥ 130	w ≥ 2000	Normal
105 < w < 130	1700 < w < 2000	Sand-Lightweight
w ≤ 105	w ≤ 1700	All-Lightweight

Table 2-1 - Default Concrete Types

Once the compressive strength of concrete f´c is input, various parameters are set to their default values.

The modulus of elasticity is computed as:[11]

$$E_c = 33w^{1.5}\sqrt{f'_c} \qquad \text{Eq. 2-10}$$

Where

[11] ACI 318-99, 8.5.1; ACI 318-02, 8.5.1; CSA-A23.3-94, 8.6.2.2

w = the density of the concrete.

For CSA A23.3

$$E_c = \left(3300\sqrt{f'_c} + 6900\right)\left(\frac{\gamma}{2300}\right)^{1.5} \qquad \text{Eq. 2-11}$$

Where

γ = the unit weight of concrete.

The square root of f'_c is limited to for the computation of shear strength provided by concrete, V_c, and development lengths.[12]

$$f_{ct} = 6.7\sqrt{f'_c} \qquad \text{Eq. 2-12}$$

Eq. 2-10 and Eq. 2-12 are used internally and cannot be modified.

The average split tensile strength is used to compute the modulus of rupture and reinforcement development lengths. For normal weight concrete, the default value of f_{ct} used by **pcaSlab** is set equal to:[13]

$$f_{ct} = 6.7\sqrt{f'_c} \qquad \text{Eq. 2-13}$$

In no case will $f_{ct}/6.7$ exceed 100 psi. The value f_{ct} will be modified according to the concrete type. Table 2-2 shows the default values for f_{ct} for different concrete types. No interpolation is performed for partial sand replacement.

[12] ACI 318-99, 11.1.2; ACI 318-02, 11.1.2

[13] ACI 318-99, 9.5.2.3; ACI 318-02, 9.5.2.3

Type	f_{ct}
Normal	$6.7\sqrt{f_c^{'}}$
Sand-Lightweight	$(0.85)\,6.7\sqrt{f_c^{'}}$
All-Lightweight	$(0.75)\,6.7\sqrt{f_c^{'}}$

Table 2-2 Default Average Splitting Tensile Strength

The modulus of rupture is used to determine the cracking moment when computing the effective moment of inertia for deflection computations. The default value of f_r, modulus of rupture, is set equal to:

For ACI 318

$$f_r = 1.12 f_{ct} \qquad \text{Eq. 2-14}$$

If f_{ct} is that given in Eq. 2-13, Eq. 2-14 is equal to:[14]

$$f_r = 7.5\sqrt{f_c^{'}} \qquad \text{Eq. 2-15}$$

For CSA A23.3

$$f_r = 7.5\sqrt{f_c^{'}} \qquad \text{Eq. 2-16}$$

for beams and columns and

$$f_r = \frac{0.6\sqrt{f_c^{'}}}{2} \qquad \text{Eq. 2-17}$$

for slabs.

[14] ACI 318-99, 9.5.2.3; ACI 318-02, 9.5.2.3; CSA A23.3-94, 8.6.4 & 13.3.6

There is no limit imposed on f_r. Entering a large value of f_r will produce deflections based on gross properties, (i.e., uncracked sections).

The default values for the longitudinal reinforcement yield strength, f_y, and shear reinforcement yield strength, f_{yv}, if applicable, are set equal to 60 ksi (413 MPa) for ACI and 400 MPa for CSA.

The Equivalent Frame Method

The equivalent frame method, as described in the Code,[15] is used by **pcaSlab** for both analysis and design.

The Code specifies procedures for the analysis and design of slab systems reinforced for flexure in more than one direction, with or without beams between the supports. A two-way slab[16] system, including the slab and its supporting beams, columns, and walls may be designed by either of the following procedures:

- The Direct Design Method
- The Equivalent Frame Method

pcaSlab uses the Equivalent Frame Method of analysis. It should be noted that this method is based on extensive analytical and experimental studies conducted at the University of Illinois. Note also that there are no restrictions on the number of slab spans or on dead-to-live load ratios in this method of analysis.

The first step in the frame analysis is to divide the three-dimensional building into a series of two-dimensional frames extending to the full height of the building. Horizontal members for each frame are formed by slab strips as shown in Figure 2-5. For vertical loads, each story (floor and/or roof) may be analyzed separately with the supporting columns being considered fixed at their remote ends. (Figure 2-6).

[15] ACI 318-99, Chapter 13; ACI 318-02, Chapter 13; CSA A23.3-94, Clause 13

[16] Implies a slab supported by isolated supports which permits the slab to deflect in two orthogonal directions.

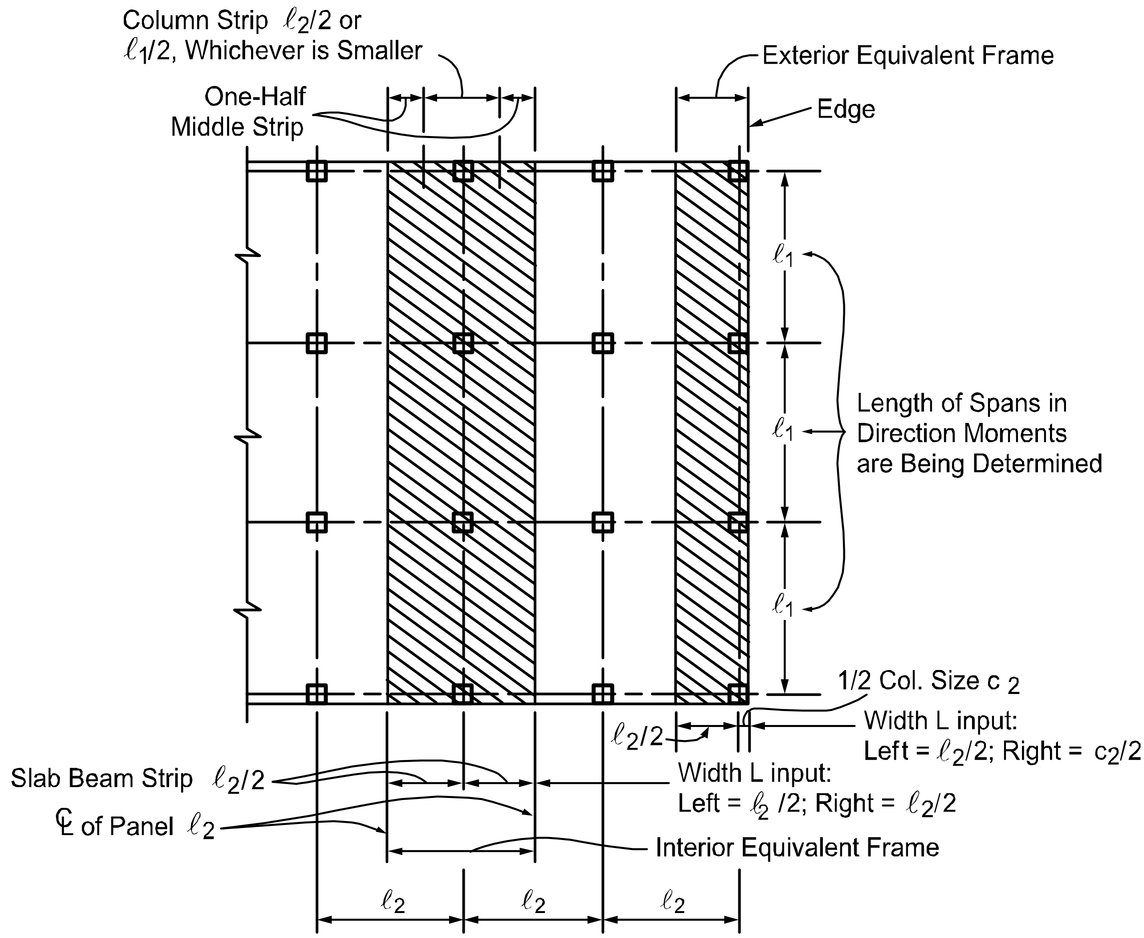


Figure 2-5 Design strips

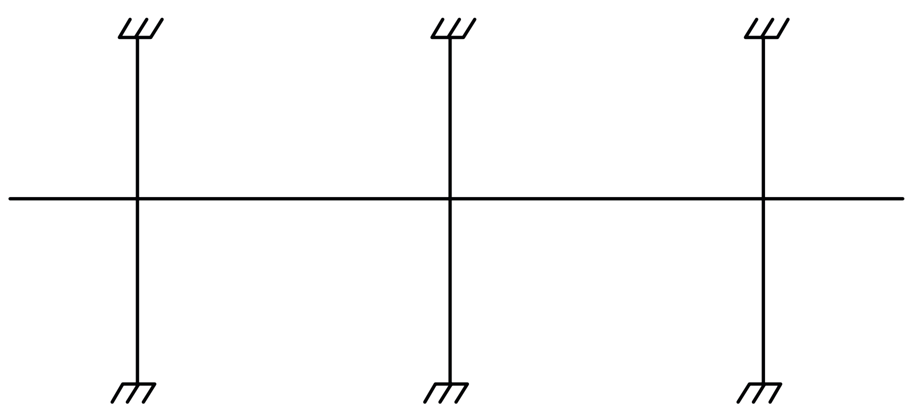


Figure 2-6 Analytical model for vertical loads for a typical story

Stiffness Characteristics

The stiffness factors for the horizontal members (the slab beams) and the vertical members (the equivalent columns) are determined using segmental approach.

Slab Beams

The moment of inertia of the slab beam elements *between* the faces of the columns (or column capitals) is based on the uncracked section of the concrete including beams or drop panels. The moment of inertia *from* the face of the column (or capital) to the centerline of the column (or capital) is considered *finite* and is dependent on the transverse dimensions of the panel and support. This reduced stiffness (as compared to the infinite stiffness assumed in previous codes) is intended to soften the slab at the joint to account for the flexibility of the slab away from the support. This is consistent with provisions of the Code.[17]

Figure 2-7 shows the changes in stiffness between a slab, and a drop panel, and a column (or capital).

Columns

The computation of the column stiffness is more complicated as it utilizes the concept of an equivalent column. Theoretical slab studies have shown that the positive moment in a slab may increase under pattern loads, even if rigid columns are used, because of the flexibility of the slab away from the column. However, if a two-dimensional frame analysis is applied to a structure with rigid columns, pattern loads will have little effect. To account for this difference in behavior between slab structures and frames, the equivalent column torsional member, as shown in Figure 2-8, runs transverse to the direction in which the moments are being determined. The transverse slab beam can rotate even though the column may be infinitely stiff, thus permitting moment distribution between adjacent panels. It is seen that the stiffness of the equivalent column is affected by both the flexural stiffness of the columns and the torsional stiffness of the slabs or beams framing into the columns. Note that the method of computation of column stiffness is in accordance with the requirements of the Code[18] Figure 2-9 shows a schematic representation of the stiffness of typical columns.

The column stiffness is based on the column height, l_c, measured from mid-depth of the slab above, to the mid-depth of the slab below. **pcaSlab** calculates the stiffness of the column below the design slab, taking into account the design slab system at its top end.

---

[17] ACI 318-99, 13.7.3; ACI 318-02, 13.7.3; CSA-A23.3-94, 13.9.2.3

[18] ACI 318-99, 13.7.4; ACI 318-02, 13.7.4; CSA-A23.3-94, 13.9

pcaSlab calculates the stiffness of the column above the design slab taking only the slab depth into account at its top end; column capitals, beams, or drops are ignored.

The computation of the torsional stiffness of the member requires several simplifying assumptions. The first step is to assume dimensions of the transverse torsional slab-beam members. Assumptions for dimensions of typical torsional members are shown in Figure 2-10.

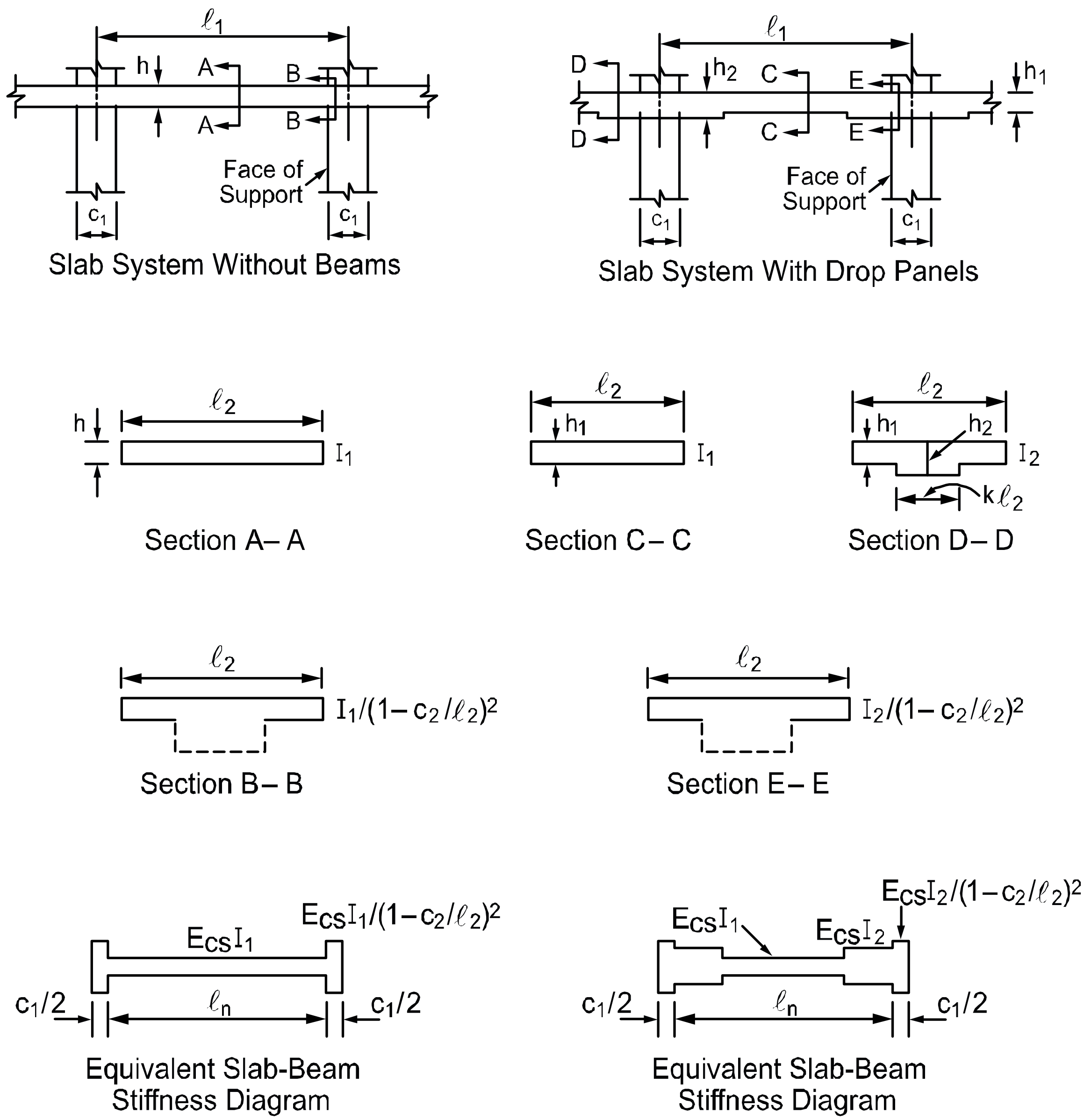


Figure 2-7 Sections for calculating slab-beam stiffness, Ksb

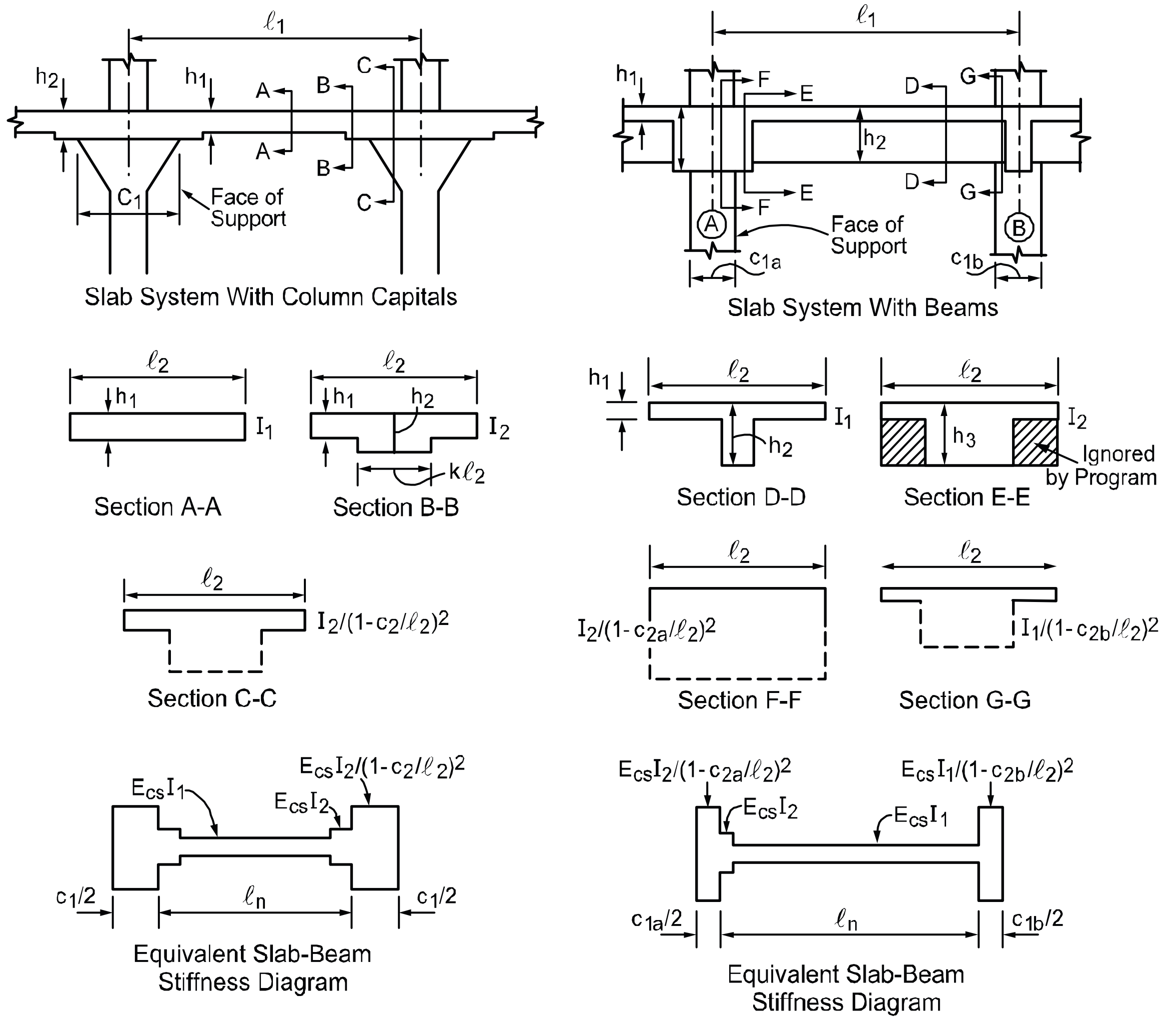


Figure 2-7 continued

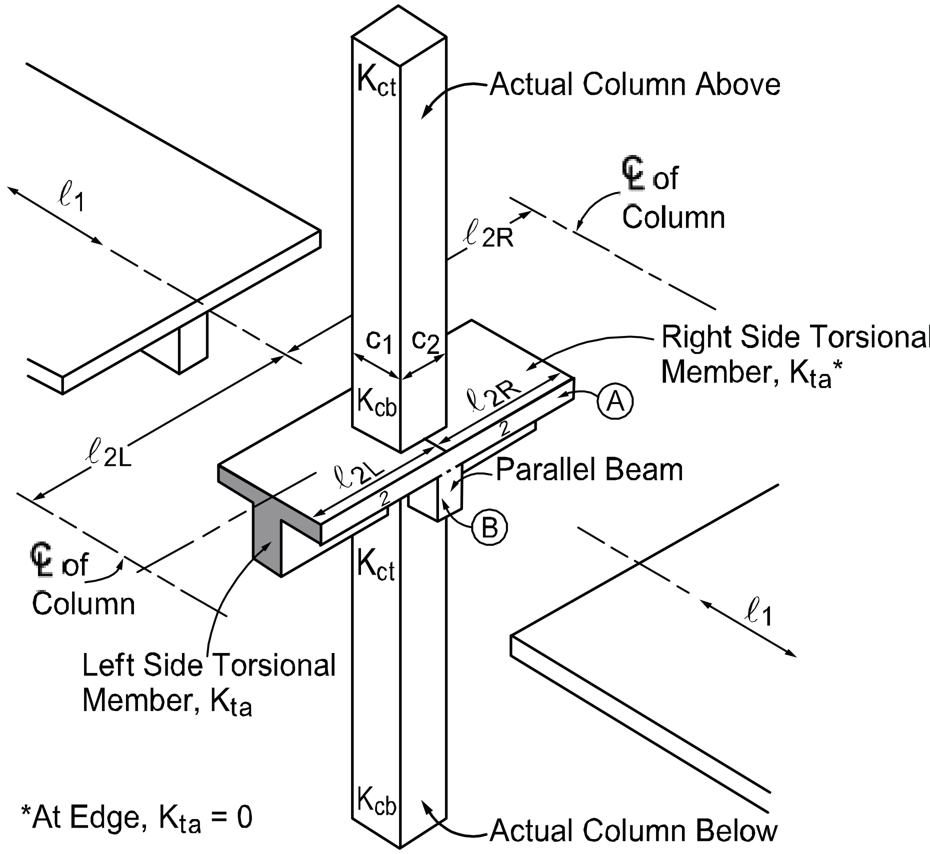


Figure 2-8 The equivalent column

The stiffness, K_t of the torsional member is given by the following expression:[19]

$$K_t = \sum \frac{9E_{CS}C}{l_2\left(1 - \frac{c_2}{l_2}\right)^3} \qquad \text{Eq. 2-18}$$

Where

E_{cs} = modulus of elasticity for slab concrete.

C = cross-sectional constant defining torsional properties; see Eq. 2-19. It is a conservatively low approximation of the torsional rigidity of rectangular sections

[19] ACI 318-99, 13.7.5; ACI 318-02, 13.7.5; CSA-A23.3-94, Eq. 13.13

when assuming elastic behavior. For the CSA Standard l_2 is taken as the smaller of l_2 or l_1.

c_2 = size of rectangular column or capital measured transverse to the direction in which moments are being determined.

l_2 = For ACI 318 length of span transverse to l_1, measured on each side of the column. For CSA A23.3-94 l_2 is taken as the smaller of l_2 or l_1.

The constant C is evaluated for the cross section by dividing it into separate rectangular parts and by carrying out the following summation:[20]

$$C = \sum \left(1 - 0.63\frac{x}{y}\right)\frac{x^3 y}{3} \qquad \text{Eq. 2-19}$$

Where

x = Short overall dimension of the rectangular part of a cross section

y = Long overall dimension of the rectangular part of a cross section.

As a result of Eq. 2-19, walls[21] running the full width of a slab ($c_2 = l_2$) cannot be modeled by the Equivalent Frame Method.

When beams frame into the column in the direction of analysis, the value of K_t as computed in Eq. 2-18 is multiplied by the ratio of the moment of inertia of the slab with the beam (I_{sb}) to the moment of inertia of the slab without the beam (I_s), as shown:

$$K_{ta} = K_t \frac{I_{sb}}{I_s} \qquad \text{Eq. 2-20}$$

With reference to Figure 2-8, I_s is computed from part A (slab without beam), whereas I_{sb} is computed from both parts A and B (slab with beam).

[20] CSA-A23.3-94, Eq. 13-14

[21] Instead walls can be modeled as long supports less than the full design width of the slab. To obtain a uniform distribution of the end moment along the column and middle strips, the width of the wall must be greater than 75% of the design strip.

Knowing the column stiffness, K_c, and the stiffness of the attached torsional member, K_t, the stiffness of the equivalent column, K_{ec}, is computed from the equation:

$$K_{ec} = \frac{K_{ct} + K_{cb}}{1 + \dfrac{K_{ct} + K_{cb}}{K_{ta} + K_{ta}}} \qquad \text{Eq. 2-21}$$

Where

K_{ct}, K_{cb} = top and bottom column stiffness,

K_{ta} = the stiffness of the left and right torsional member.

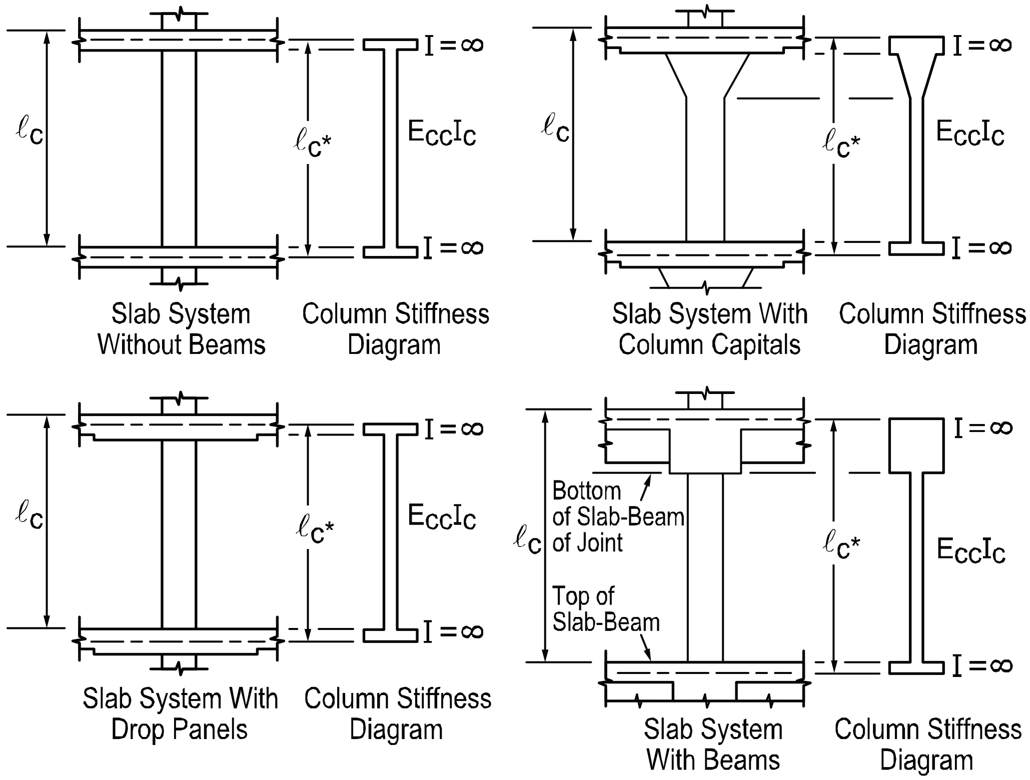
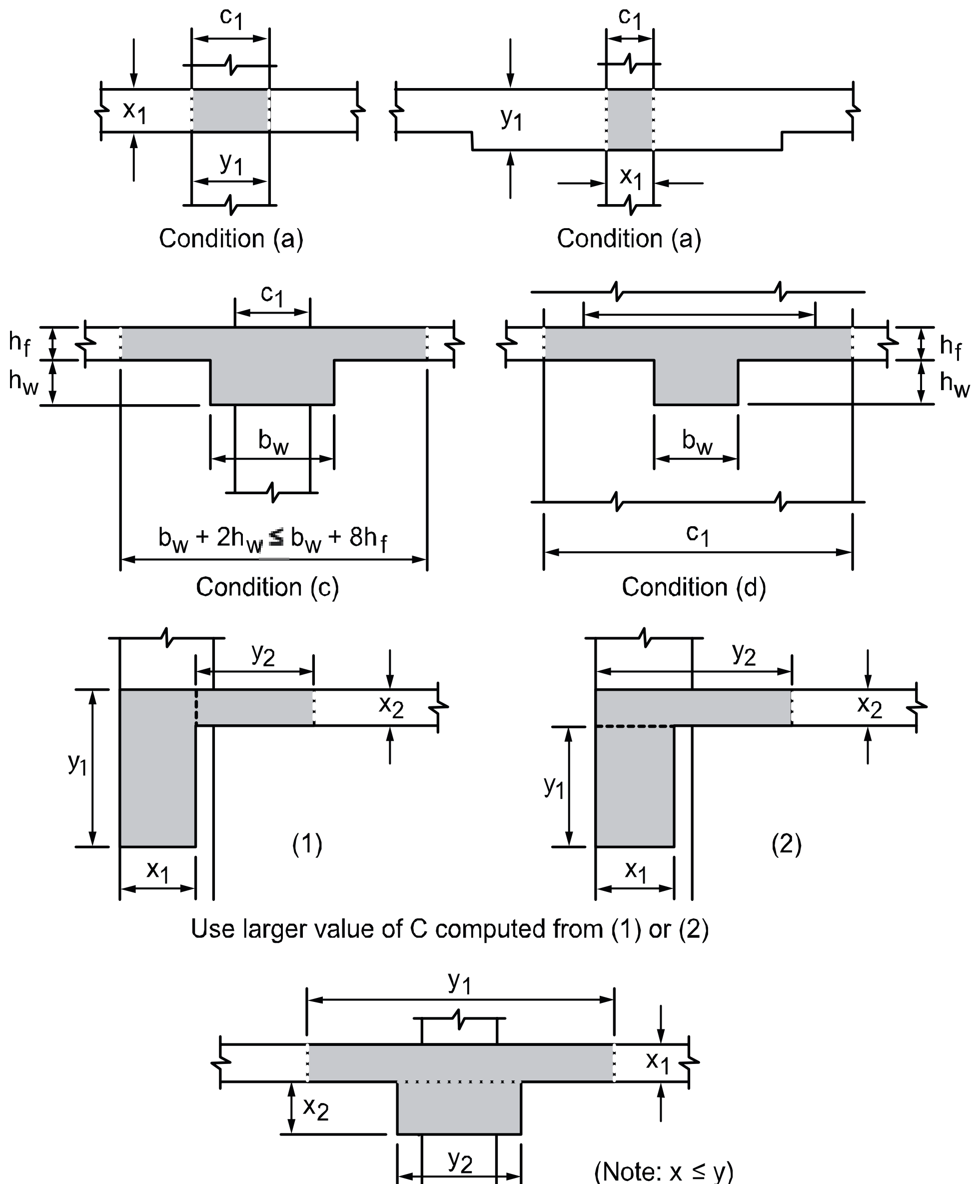


Figure 2-9 Sections for calculating the stiffness (Kc) of the column below the design floor (lc-input, lc*-computed)



$$C = \sum\left[\left(1-0.63\,\frac{x_1}{y_1}\right)\frac{x_1^3 y_1}{3}\right] + \left[\left(1-0.63\,\frac{x_2}{y_2}\right)\frac{x_2^3 y_2}{3}\right]$$

Figure 2-10 Section of the attached torsional members

Loading

All applied loads are input as unfactored loads. There are no limitations imposed on the ratio of dead to live loads in the Equivalent Frame Method. Results of gravity load and lateral load analyses may be combined, however, the effects of cracking and reinforcement on stiffness must be accounted for in the lateral load analysis.

Self-Weight

The self-weight of the floor system is computed internally by **pcaSlab**. The weights of the slabs, drops, and longitudinal and transverse beams are considered in the self weight computations. Only the concrete weight is considered, the reinforcement weight is ignored. The weight of longitudinal beams is ignored starting at the column centerline, for a length equal to one-half c_1, the column dimension in the direction of analysis. This will produce slightly less self-weight than actually present for beams wider than c_2, the column's transverse dimension.

Superimposed Loading

All superimposed vertical loading is considered to act over the entire transverse width of the slab. For slab systems with beams, loads supported directly by the beam (such as the weight of the beam stem or a wall supported directly by the beams) are also assumed to be distributed over the entire transverse width of the strip. An additional analysis may be required, with the beam section designed to carry these loads in addition to the portion of the slab moments assigned to the beam.

Lateral Loading

For lateral loads, each frame should be analyzed as a unit for the entire height of the building (Figure 2-11). Computer programs, such as **PCA-Frame**, are available for performing such analyses. It should be realized that, for lateral load analysis, slab-beam elements may have a reduced stiffness due to cracking as well as other assumptions made for the effective slab width used for the lateral analysis. The moments obtained from such an analysis may then be input into the equivalent frame model using **pcaSlab** to determine the appropriate design moments under combined vertical and lateral loads.

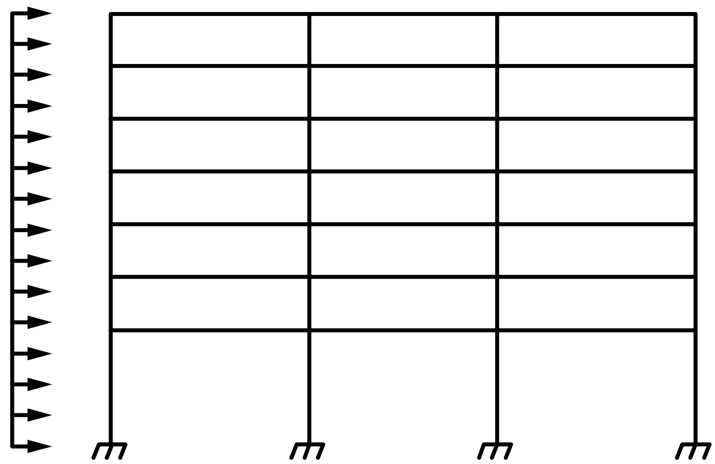


Figure 2-11 Analytical model for lateral loads

By default, **pcaSlab** distributes the effect of lateral load moment, the difference between the total load moments, vertical plus lateral, and the vertical only moment envelopes, to the column strip and middle strip according to the code distribution factors computed for vertical loads (see Table 2-3 through Table 2-5 later in this chapter).

Loading Patterns

The analysis of floor systems requires the consideration of several loading configurations. For example, the two adjacent spans loaded may produce the maximum shear stress around a column, while the alternate spans loaded may produce the maximum flexural moments. To cover different loading scenarios **pcaSlab** generates live load case based on the following load patterns (Figure 2-12):

Loading Pattern No. I: All spans loaded with live load (L_{ALL}).

Loading Pattern No. 2: Alternate (odd) spans loaded with live load (L_{ODD}).

Loading Pattern No. 3: Alternate (even) spans loaded with live load (L_{EVEN}).

Loading Pattern No. 4: Two adjacent spans loaded with live load (L_{SUPP}).

pcaSlab reduces the magnitude of live load patterns No.2 through No.4 by a predefined ratio. The default live load pattern ratio selected by the program equals 75% as permitted by the code.[22] The user has the ability to select different value for the pattern

[22] ACI 318-99, 13.7.6.3; ACI 318-02, 13.7.6.3; CSA-A23.3-94, 13.9.4

ratio within the range 0-100%. The pattern No.1 with all spans loaded is always considered with full unreduced magnitude.

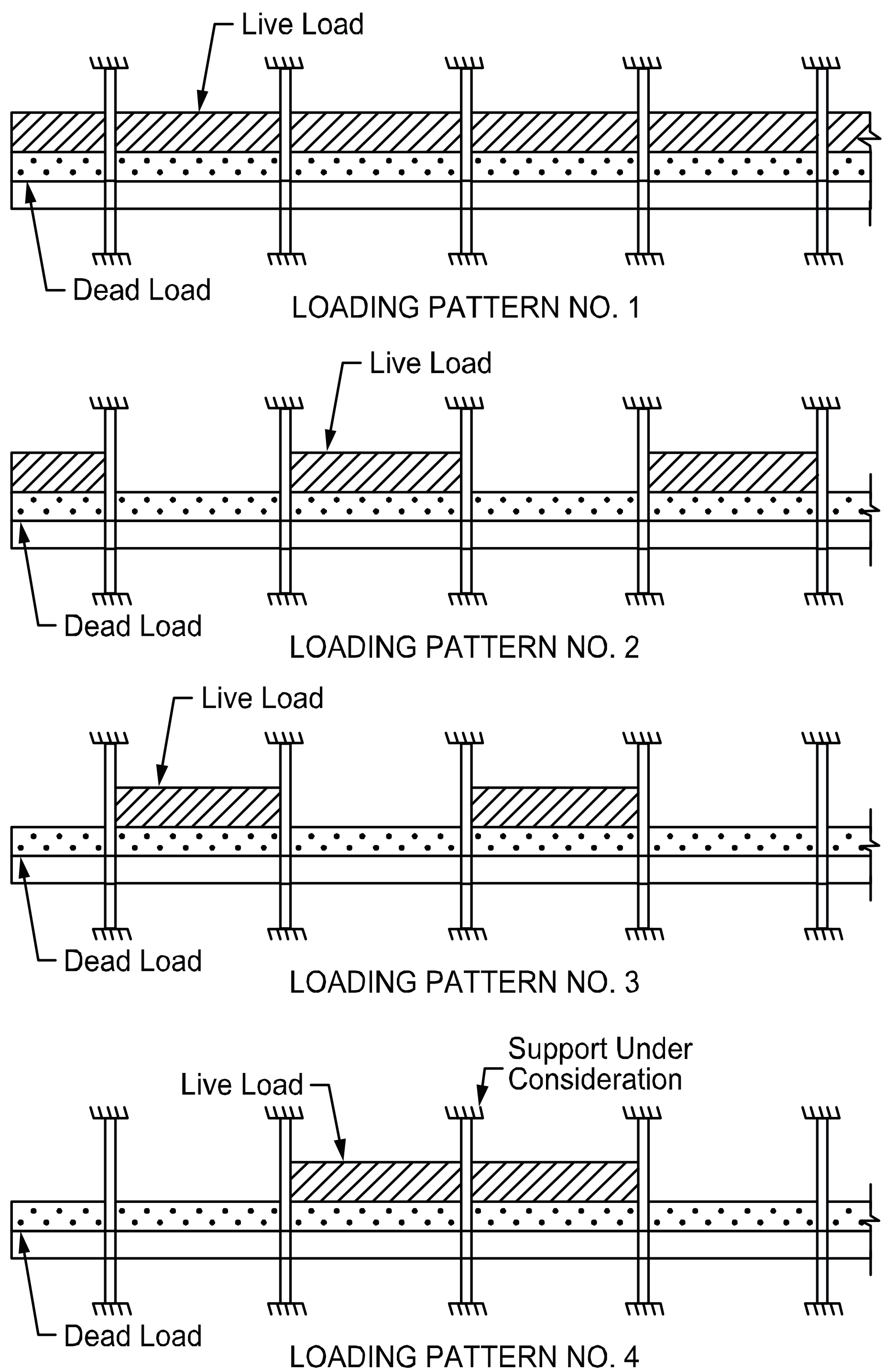


Figure 2-12 Basic pattern live loads

Load Combinations

pcaSlab allows defining up to 20 load combinations. The user has full control over the combinations. The program contains predefined (build into the program) default primary load combinations for the supported codes. These default combinations are created when starting a new project.

For the ACI 318-99 Code, the default combinations of the Dead (D), Live (L), Wind (W) and Earthquake (E) loads considered by the program are:

U1 = 1.4D + 1.7L

U2 = 0.75(1.4D + 1.7L + 1.7W)

U3 = 0.75(1.4D + 1.7L -- 1.7W)

U4 = 0.75(1.4D + 1.7W)

U5 = 0.75(1.4D -- 1.7W)

U6 = 0.9D + 1.3W

U7 = 0.9D -- 1.3W

U8 = 0.75(1.4D + 1.7L + 1.7*1.1E)

U9 = 0.75(1.4D + 1.7L -- 1.7*1.1E)

U10 = 0.75(1.4D + 1.7*1.1E)

U11 = 0.75(1.4D -- 1.7*1.1E)

U12 = 0.9D + 1.43E

U13 = 0.9D -- 1.43E

For the ACI 318-02 Code, the default combinations of the Dead (D), Live (L), Wind (W) and Earthquake (E) loads considered by the program are:

U1 = 1.4D

U2 = 1.2D + 1.6L

U3 = 1.2D + 1.6L + 0.8W

U4 = 1.2D + 1.6L – 0.8W

U5 = 1.2D + 1.0L + 1.6W

U6 = 1.2D + 1.0L – 1.6W

U7 = 0.9D + 1.6W

U8 = 0.9D -- 1.6W

U9 = 1.2D + 1.0L + 1.0W

U10 = 1.2D + 1.0L – 1.0W

U11 = 0.9D + 1.0W

U12 = 0.9D -- 1.0W

For the CSA-A23.3 Code, the default combinations of the Dead (D), Live (L), Wind (W) and Earthquake (E) loads considered by the program are:

U1 = 1.25D + 1.5L

U2 = 1.25D + 0.7(1.5L + 1.5W)

U3 = 1.25D + 0.7(1.5L – 1.5W)

U4 = 0.85D + 1.5W

U5 = 0.85D – 1.5W

U6 = 1.0D + 0.5L + 1.0E

U7 = 1.0D + 0.5L – 1.0E

U8 = 1.0D + 1.0E

U9 = 1.0D – 1.0E

Column and Middle Strip Widths

The Code[23] defines the width of the column strip on each side of the column centerline as being one-fourth of the smaller of either the transverse or the longitudinal span. These widths are printed as part of the echo of input data.

The strip widths at a support are computed by (see Figure 2-13)

---

[23] ACI 318-99, 13.2.1; ACI 318-02, 13.2.1; CSA-A23.3-94, 13.1

Column strip

$$W_{CS} = \min\left\{\begin{array}{l} \min\left\{\frac{l_{2,l}}{2}, \frac{l_1}{4}\right\}_i + \min\left\{\frac{l_{2,r}}{2}, \frac{l_1}{4}\right\}_i \\ \min\left\{\frac{l_{2,l}}{2}, \frac{l_1}{4}\right\}_{i+1} + \min\left\{\frac{l_{2,r}}{2}, \frac{l_1}{4}\right\}_{i+1} \end{array}\right\} \qquad \text{Eq. 2-22}$$

Middle strip

$$W_{MS} = \min\{l_{2,i}, l_{2,i+1}\} - W_{CS} \qquad \text{Eq. 2-23}$$

The strip widths in the span are defined as (see Figure 2-14):

$$W_{CS} = \min\left\{\frac{l_{2,l}}{2}, \frac{l_1}{4}\right\} + \min\left\{\frac{l_{2,r}}{2}, \frac{l_1}{4}\right\} \qquad \text{Eq. 2-24}$$

Middle strip

$$W_{MS} = l_2 - W_{CS} \qquad \text{Eq. 2-25}$$

Where

min{a, b} = the smaller of “a” and “b”.

l_1 = span length in the direction of analysis

l_2 = the total input transverse strip width

$l_{2,1}$, $l_{2,r}$ = the input transverse strip widths on the left and right of column centerline, respectively

i, i+1 = span left and right of column centerline, respectively.

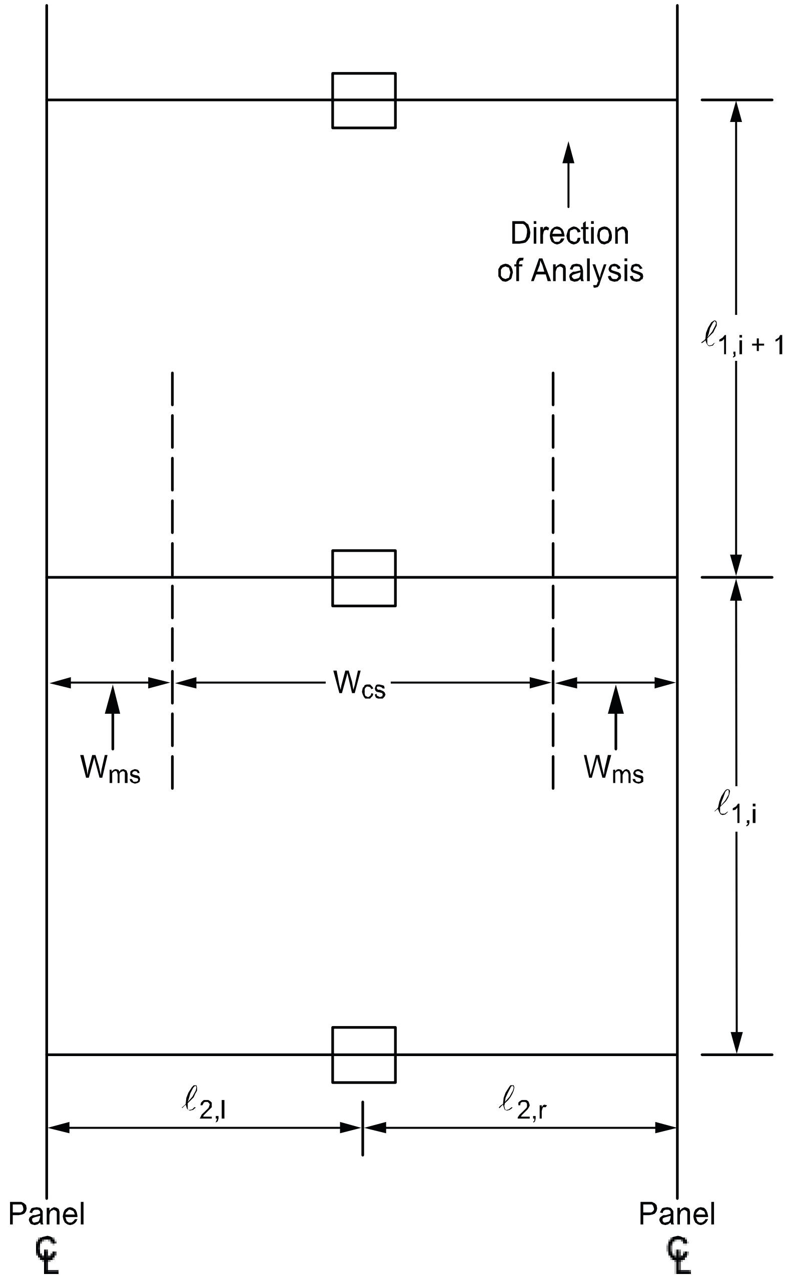


Figure 2-13 Strips widths at support

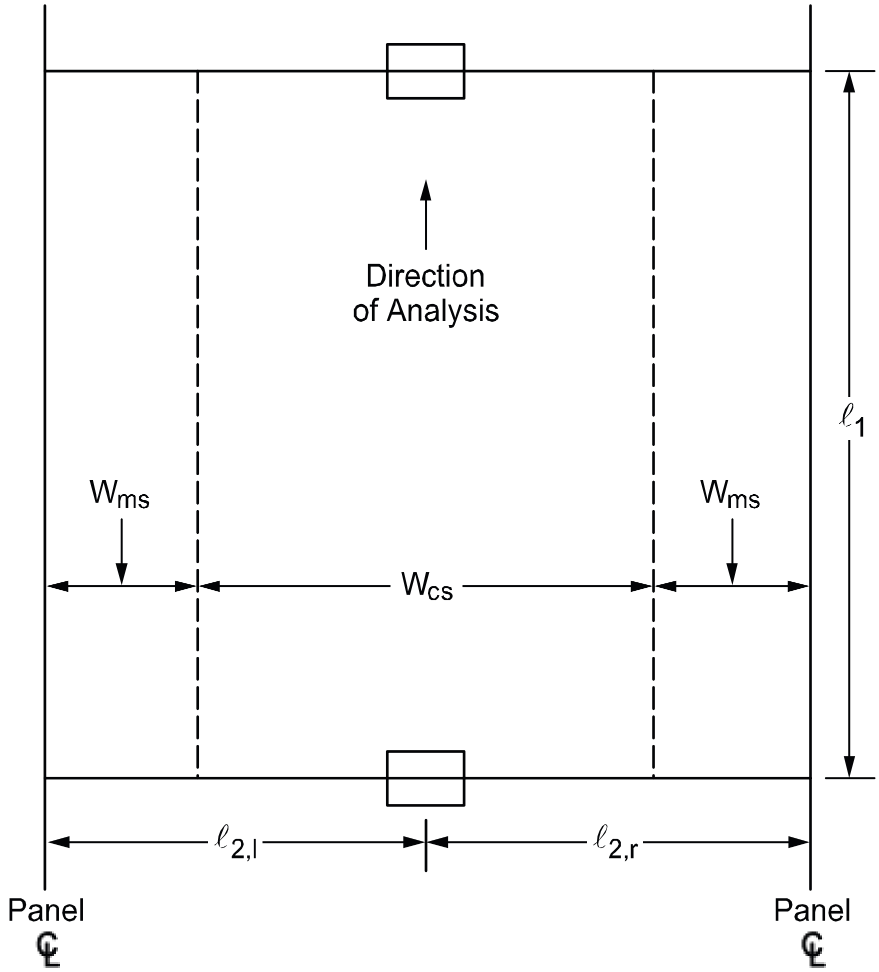


Figure 2-14 Strip widths in span

If a beam exists

$$W_{CS} = W_{CS} - W_b \quad \text{Eq. 2-26}$$

If the beam width is greater than the column strip width, W_{cs}, then

$$\begin{aligned} W_{MS} &= W_{MS} - (W_b - W_{CS}) \\ W_{CS} &= 0 \end{aligned} \quad \text{Eq. 2-27}$$

Where

W_b = beam width.

For exterior frames, the edge width should be specified to the edge of the slab from the column centerline.

Design Moments

pcaSlab considers negative moments for design purposes as those producing tension at the top of the slab. The negative design moment is taken at a section located at the face of the column, or column capital, but in no case is it considered at a location greater than 0.175 of the longitudinal span length, l_1, away from the center of the column.[24] This imposes a limit on long narrow supports, in order to prevent undue reduction in the design moment. For slab systems with transverse beams, the face of a beam is not considered as the face of support. For end columns with capitals, the moments are taken at the midpoint of the capital extension.[25] The column and middle strip moments correspond to the moments assigned to the slab element only.

The column strips are proportioned to resist the portions in percent of interior negative factored moments according to Table 2-3.[26]

l_2/l_1	0.5	1.0	2.0
$(\alpha_1 l_2/l_1) = 0$	75	75	75
$(\alpha_1 l_2/l_1) \geq 1.0$	90	75	45

Table 2-3 - Column Strip Percent of Interior Negative Factored Moments at Supports

The column strips are proportioned to resist the portions in percent of exterior negative factored moments according to Table 2-4.[27]

l_2/l_1		0.5	1.0	2.0
$(\alpha_1 l_2/l_1) = 0$	$\beta_t = 0$	100	100	100
	$\beta_t \geq 2.5$	75	75	75
$(\alpha_1 l_2/l_1) \geq 1.0$	$\beta_t = 0$	100	100	100
	$\beta_t \geq 2.5$	90	75	45

Table 2-4 - Column Strip Percent of Exterior Negative Factored Moments at Supports

[24] ACI 318-99, 13.7.7.1; ACI 318-02, 13.7.7.1; CSA3-A23.3-94, 13.9.5.1

[25] ACI 318-99, 13.7.7.2; ACI 318-02, 13.7.7.2; CSA3-A23.3-94, 13.9.5.2

[26] ACI 318-99, 13.6.4.1; ACI 318-02, 13.6.4.1; CSA3-A23.3-94, 13.9.5.1

[27] ACI 318-99, 13.6.4.2; ACI 318-02, 13.6.4.2

The values α_1 in Table 2-3 and Table 2-4 and β_t in Table 2-4 are defined as:

α_1 = ratio of flexural stiffness of the beam section to flexural stiffness of a width of slab bounded by centerlines of adjacent panels (if any) on each side of the beam in the direction of analysis. For flat plates, flat slabs, and waffle $\alpha_1 l_2/l_1 = 0$

β_t = ratio of torsional stiffness of an edge beam section to flexural stiffness of a width of slab equal to the span length of the beam, center-to-center of supports;[28] see Eq. 2-28. When no transverse beams are present, $\beta_t = 0$.

$$\beta_t = \frac{E_{cb}C}{2E_{CS}I_S} \qquad \text{Eq. 2-28}$$

Where

E_{cb} = modulus of elasticity of beam concrete.

E_{cs} = modulus of elasticity of slab concrete.

C = cross-sectional constant to define torsional properties; see Eq. 2-19

I_s = moment of inertia of the gross section of the slab about its centroidal axis.

For intermediate values of (l_2/l_1), $(\alpha_1 l_2/l_1)$ and β_t the values in Table 2-3 and Table 2-4 are interpolated using equations Eq. 2-29 and Eq. 2-30.

Percentage of negative factored moment at interior support to be resisted by column strip:

$$= 75 + 30\left(\frac{\alpha_1 l_2}{l_1}\right)\left(1 - \frac{l_2}{l_1}\right) \qquad \text{Eq. 2-29}$$

Percentage of negative factored moment at exterior support to be resisted by column strip:

$$= 100 - 10\beta_t + 12\beta_t\left(\frac{\alpha_1 l_2}{l_1}\right)\left(1 - \frac{l_2}{l_1}\right) \qquad \text{Eq. 2-30}$$

When a column width, c_2, is equal to or greater than 75 percent of the strip width, l_2 the negative moment is uniformly distributed across l_2.[29]

---

[28] ACI 318-99, 13.0; ACI 318-02, 13.0; CSA3-A23.3-94, 13.0

[29] ACI 318-99, 13.6.5; ACI 318-02, 13.6.5

When designing by the CSA A23.3-94 Code, a portion of the total positive or interior negative moment equivalent to:

$$\frac{\alpha_1}{\left[1+\left(\frac{l_2}{l_1}\right)^2\right]} \qquad \text{Eq. 2-31}$$

is resisted by the beam. For exterior supports the beam is proportioned to resist 100% of the negative moment.

That portion of the moment not resisted by the beam is resisted by the slab. The reinforcement required to resist this moment is distributed evenly across the slab.

For ACI designs the longitudinal beams are proportioned to resist 85 percent of the column strip moments if $\alpha_1 l_2/l_1$ is equal to or greater than 1.0. For values of $\alpha_1 l_2/l_1$ between 0 and 1.0, the beam is designed to resist a proportionate percentage of the column strip moment between 0 and 85.[30]

When lateral loads are present, **pcaSlab**, by default, distributes the effects of lateral loads according to Table 2-3 and Table 2-4.

The middle strips are proportioned to resist the portion of the total factored moments that is not resisted by the column strips.

For design purposes, **pcaSlab** computes the amount of reinforcement for the moments on the left and right sides of the support. The negative design moment is the moment which requires the most area of reinforcement to be resisted. The location, left or right of the support, of the maximum moment may vary when systems differ on each side of the support (for example, a system with beams on one side only). **pcaSlab** considers positive moments for design purposes as those producing tension at the bottom of the slab. The column strips are proportioned to resist the portions in percent of positive factored moments according to Table 2-5.[31]

For ACI designs the column strips are proportioned to resist the portions in percent of factored vertical load moments along the span according to Table 2-5.[31]

---

[30] ACI 318-99, 13.6.5; ACI 318-02, 13.6.5

[31] ACI 318-99, 13.6.4.4; ACI 318-02, 13.6.4.4

l_2/l_1	0.5	1.0	2.0
$(\alpha_1 l_2/l_1) = 0$	60	60	60
$(\alpha_1 l_2/l_1) \geq 1.0$	90	75	45

Table 2-5 - Column Strip Percent of Positive Factored Moments

For intermediate values of (l_2/l_1) and ($\alpha_1 l_2/l_1$) the values in Table 2-5 are interpolated using Eq. 2-32 as follows:

$$= 60 + 30\left(\frac{\alpha_1 l_2}{l_1}\right)\left(1.5 - \frac{l_2}{l_1}\right) \qquad \text{Eq. 2-32}$$

The middle strips are proportioned to resist the remainder of the total factored moments.

Note, for flat plates, flat slabs, and waffle slabs, $\alpha_1 l_2/l_1 = 0$.

Shear Analysis of Slabs

Three types of shear can occur on slab systems: wide beam shear, punching shear, and moment transfer shear. **pcaSlab** checks wide beam shear at a critical section located at a distance equal to the effective depth away from the face of the supports. The tributary area for wide beam shear calculations extends from the critical section to the center of the span. For punching shear including moment transfer shear it is assumed that both types of shear act on the same critical section. The critical section for shear is defined in the Code.[32]

Figure 2-15 shows the general punching shear area used by **pcaSlab**. Note that the shaded area represents the general case and is modified for special considerations as explained below.

[32] ACI 318-99, 11.12.1.2; ACI 318-02, 11.12.1.2; CSA-A23.3-94, 13.4.3

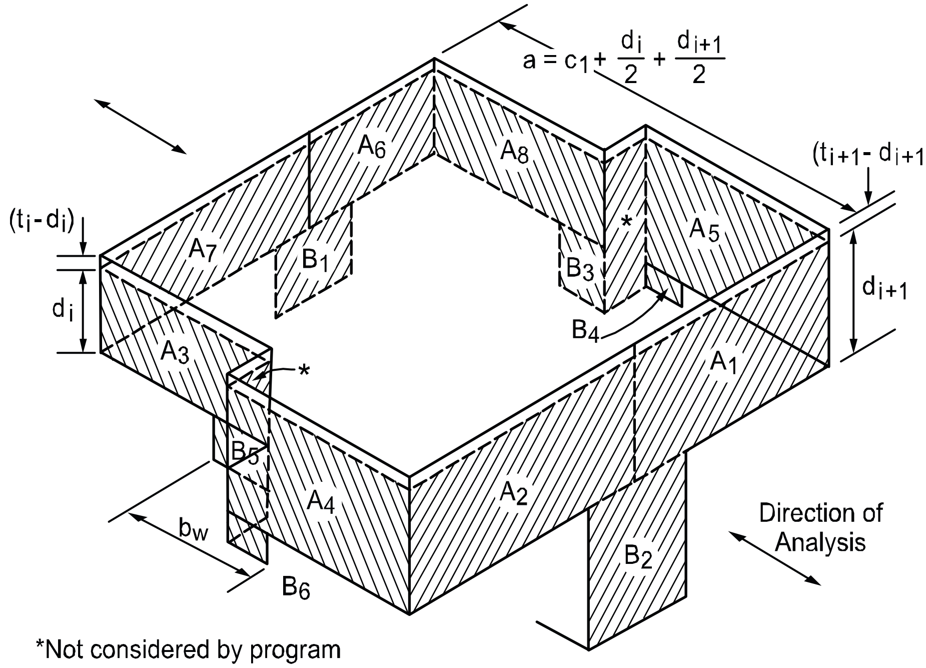


Figure 2-15 Critical section for shear at column

Shallow beams are considered in the unbalanced moment transfer as indicated in Figure 2-15 by areas B_1, B_2, B_3, B_4, B_5, B_6. Ordinarily, transverse beams transfer unbalanced moment to the column through torsion along the beam and not through shear between the slab and column. However, the Code leaves the transfer method to the engineer's judgment concerning the point at which punching shear is no longer applicable and beam shear becomes the dominate element in shear transfer to the column. In the default mode of **pcaSlab**, the program makes no such distinction and computes unbalanced moment transfer stress without regard to any beams framing into the column. It is possible, however, to have **pcaSlab** distribute the shear and torsion among the framing beams. Although the depth of the beam is considered in the critical section surfaces, the distances to the critical section are *not* increased at the intersection with any beams. This approach is conservative. **pcaSlab** does not compute torsional stresses in the slab. If in the engineer's judgment this may control, it must be computed manually.

For a circular column or column capital, a square shape with an equivalent area is assumed as shown in Figure 2-16. Critical section area for punching, A_c, is then multiplied by $\pi(D+d)/(2D\sqrt{\pi}+4d)$ to account for the difference between circular and square critical sections, where D is the diameter of a column or a column capital and "d" is the effective depth of the slab.

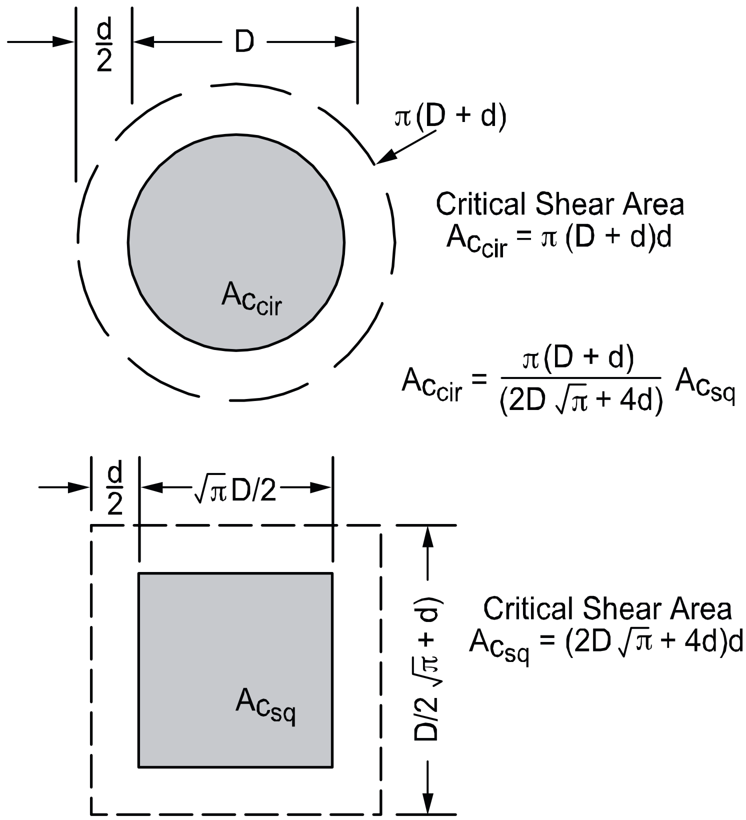


Figure 2-16 Critical section area for circular column

The critical section is considered *closed* if the concrete slab around a column extends to a distance greater than or equal to the specified treshold value. In **pcaSlab** the user may define the distance extended beyond the column face in order to consider the section *closed*. The default value for this distance is ten times the slab thickness (10h) as required by ACI code. If the critical section does not meet the distance requirement it is considered *open*.

Critical Section for Interior Supports of Interior Frames

The critical section (Figure 2-17) consists of four vertical surfaces through the slab, located at distances of d/2 beyond the support faces.

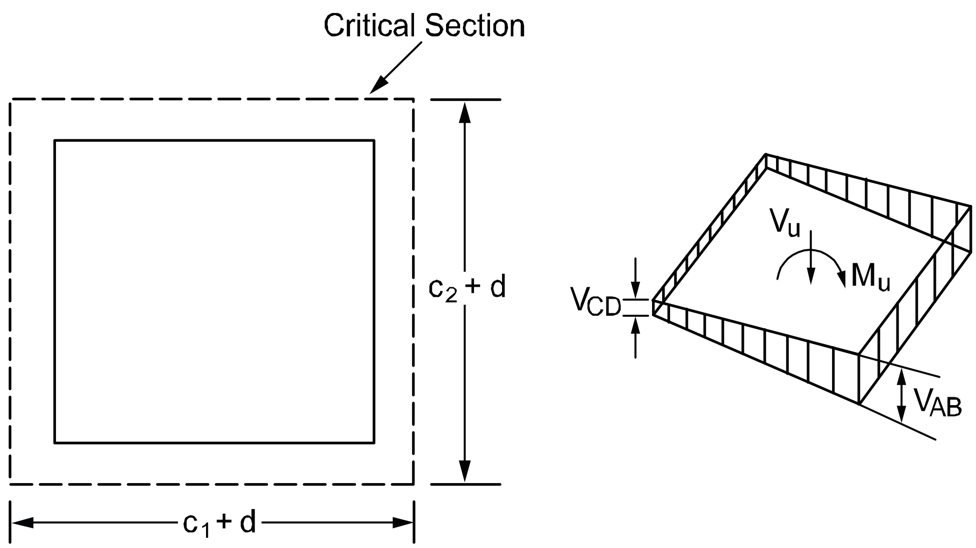


Figure 2-17 Interior supports of interior frames

The critical section for interior supports of interior frames is always closed. A closed section will have all its faces defined in Figure 2-15, resisting shear as indicated by Eq. 2-33:

$$A_C = \sum_{i=1}^{8} A_i \qquad \text{Eq. 2-33}$$

If beams frame[33] into the column, then the critical section includes the dimensions of the beams (B_1 through B_6 in Figure 2-15).

Critical Section for Exterior Supports of Interior Frames

The critical section for exterior supports of interior frames (Figure 2-18) will be either closed (full A_7 and A_6 for the first column or A_1 and A_2 for the last column in Figure 2-15) or open, depending upon the length of the cantilever in relation to slab thickness. The critical section will be considered closed when the clear cantilever span, l_c, is greater than or equal to the distance defined by the user beyond the column face (the default value is 10h). If beams frame into the column, then the critical section includes the contributions from the beam dimensions (B_1 through B_6 in Figure 2-15).

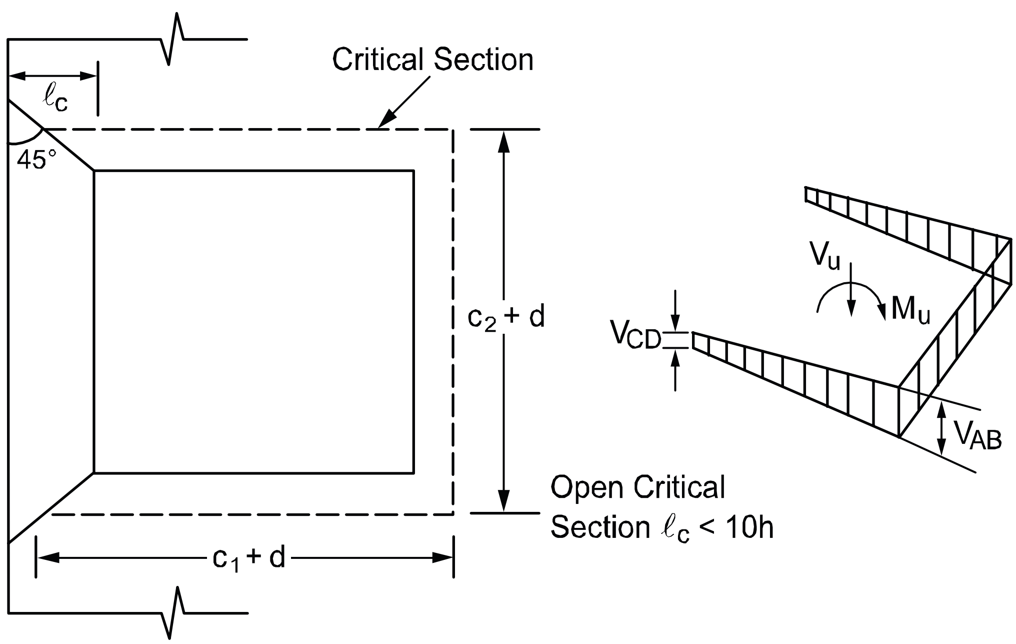


Figure 2-18 Exterior supports of interior frames

Critical Section for Interior Supports of Exterior Frames

[33] A beam is considered as framing into the column if the beam is within a face of the column.

Figure 2-19 shows the critical section for shear for an interior support of an exterior frame. Note that the section is considered as U-shaped (A_5 = 0, A_8 = 0, B_3 = 0, B_4 = 0 in Figure 2-15) and it extends up to the edge of the exterior face of the support. If beams frame into the column, then the critical section includes the contribution from the beam dimensions (B_1 through B_6 in Figure 2-15). If the exterior cantilever span is greater than or equal to the distance defined by the user beyond the column face (the default value is 10h), the section is treated as closed, that is, the support is treated as an interior support of an interior frame.

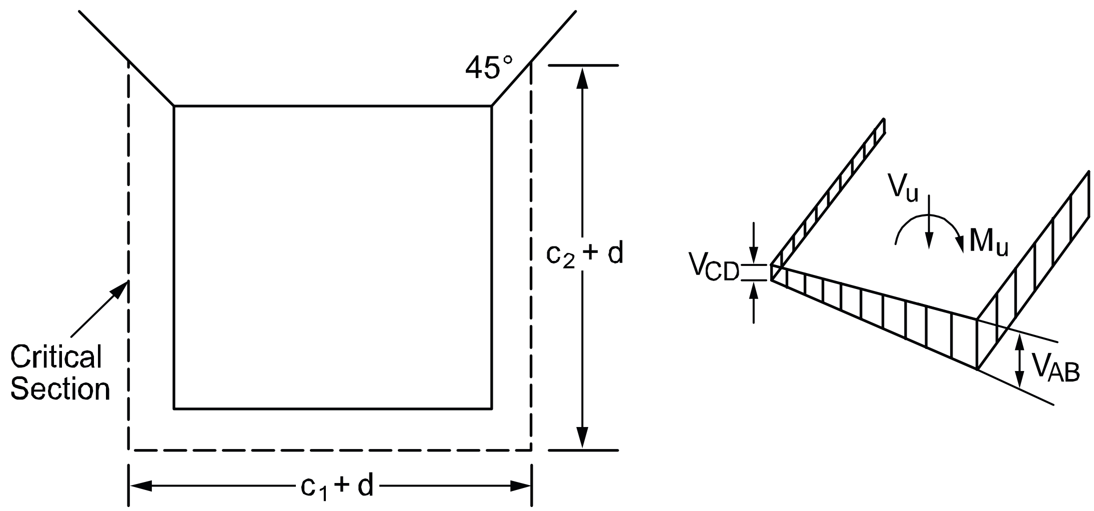


Figure 2-19 Interior supports of exterior frames

Critical Section for Exterior Supports of Exterior Frames

The critical section for an exterior support of an exterior frame will typically be L-shaped (A_5 =0, A_6 =0, A_7 =0, A_8 =0, B_1 = 0, B_3 =0, and B_4 = 0 in Figure 2-15).

If the cantilever span l_c (in the direction of analysis) is greater than or equal to the distance defined by the user beyond the column face (the default value is 10h), then the section is treated as a U-shaped interior support. If, in addition, the cantilever span in transverse direction is greater than or equal to the distance defined by the user beyond the column face (the default value is 10h), the section is treated as *closed*. If beams frame into the column, then the critical section includes the contributions from the beam dimensions.

Computation of Allowable Shear Stress at Critical Section

One-way shear strength of slabs is limited[34] to $2\sqrt{f'_c}$. Two-way shear strength of slabs is affected by concrete strength, relationship between size of loaded area and slab thickness, loaded area aspect ratio, and shear-to-moment ratio at slab-column connections.

For the ACI 318-99, ACI 318-02 and CSA A23.3-94 Code, these variables are taken into account in the allowable shear stress Eqs. 2-36 through 2-39. The allowable shear stress used by **pcaSlab** is computed at distances of d/2 around the columns and drops (if applicable) and taken as the smallest of the 3 quantities:[35]

$$v_c = \left(2 + \frac{4}{\beta_c}\right)\sqrt{f'_c} \qquad \text{Eq. 2-34}$$

$$v_c = \left(2 + \frac{\alpha_S d}{b_0}\right)\sqrt{f'_c} \qquad \text{Eq. 2-35}$$

$$v_c = 4\sqrt{f'_c} \qquad \text{Eq. 2-36}$$

Where

β_c = the ratio of the long to the short side of the column.

α_s = a constant dependent of the column location, (40 for an interior 4-sided effective critical area, 30 for an exterior 3-sided critical area, 20 for a corner 2-sided effective critical area.)

d = distance from the slab bottom to centroid of the slab tension reinforcement at support.

b_0 = the perimeter of the critical section.

The allowable shear stress around drops when waffle slabs are used is computed as:

[34] ACI 318-99, 11.3.1.1; ACI 318-02, 11.3.1.1

[35] ACI 318-99, 11.12.2.1; ACI 318-02, 11.12.2.1; CSA-A23.3-94, 13.4.4

$$\text{ACI} \quad v_c = 2\sqrt{f_c'}$$
$$\text{CSA} \quad v_c = 0.2\phi_c\sqrt{f_c'} \qquad \text{Eq. 2-37}$$

Where

ϕ_c = resistance factor for concrete.

For waffle slab systems with valid ribs defined earlier in this chapter, the allowable shear stress is increased by 10% for ACI designs.[36]

Computation of Factored Shear Force at Critical Section

The factored shear force V_u on the critical section, is by default computed as the reaction at the centroid of the critical section (e.g., column centerline for interior columns) minus the self-weight and any superimposed surface dead and live load acting within the critical section. If the section is considered *open*, two 45 degree lines are drawn from the column corners to the nearest slab edge (lines AF and DE in Figure 2-19) and the self-weight and superimposed surface dead and live loads acting on the area ADEF are omitted from V_u.

Computation of Unbalanced Moment at Critical Section

The factored unbalanced moment used for shear transfer, M_{unbal}, by default is computed as the sum of the joint moments to the left and right, taken to the centroidal axis of the critical section.

Computation of Shear Stresses at Critical Section

The punching shear stress printed by the program is based on the following:[37]

---

[36] ACI 318-99, 8.11.8; ACI 318-02, 8.11.8

[37] ACI 318-99, 11.12.6.2; ACI 318-02, 11.12.6.2; CSA-A23.3-94, 13.4.5

$$v_u = \frac{V_u}{A_c}$$ Eq. 2-38

Where:

V_u = factored shear force on the critical section described above

A_c = area of concrete, including beam if any, resisting shear transfer.

Under conditions of combined shear, V_u, and an unbalanced moment, M_{unbal}, $\gamma_v M_{unbal}$ is assumed to be transferred by eccentricity of shear about the centroidal axis of the critical section. The shear stresses printed by the program for this condition correspond to:[38]

$$v_{AB} = \frac{V_u}{A_c} + \frac{\gamma_v M_{unbal} c_{AB}}{J_c}$$ Eq. 2-39

$$v_{CD} = \frac{V_u}{A_c} - \frac{\gamma_v M_{unbal} c_{CD}}{J_c}$$ Eq. 2-40

Where

M_{unbal} = factored unbalanced moment transferred directly from slab to column, as described above.

γ_v = fraction of unbalanced moment considered transferred by the eccentricity of shear about the centroid of the assumed critical section.[39]

$$\gamma_v = (1 - \gamma_f)$$ Eq. 2-41

c_{AB}, c_{CD} = distance from centroid of critical section to face of section where stress is being computed.

Where

---

[38] ACI 318-99, 11.12.6.2; ACI 318-02, 11.12.6.2; CSA-A23.3-94, 13.4.5.5

[39] ACI 318-99, 11.12.6.2; ACI 318-02, 11.12.6.2; CSA-A23.3-94, Eq, 13-8

$$\gamma_f = \frac{1}{1+(2/3)\sqrt{b_1/b_2}} \qquad \text{Eq. 2-42}$$

J_c = property of the assumed critical section analogous to polar moment of inertia

b_1 = width of critical section in the direction of analysis

b_2 = width of the critical section in the transverse direction.

pcaSlab calculates v_u as the absolute maximum of v_{AB} and v_{CD}. Local effects of concentrated loads are not computed by **pcaSlab** and must be calculated manually.

Shear Resistance at Corner Columns

For CSA A23.3-94 Code the program performs one-way shear resistance check in the vicinity of corner columns. A critical shear section is located d/2 from the column corner. The minimum length section is selected using an optimization algorithm which analyzes sections at different angles. The extension to the cantilevered portion is considered by a length not to exceed effective slab thickness d. For slabs with edge beams or drop panels the check including the contribution of these components should be preformed manually.

Shear Analysis of Longitudinal Beams

When longitudinal beams are present in a span, **pcaSlab** will compute the shear reinforcement requirements for the beams. Table “Longitudinal Beam Shear Reinforcement Required” in the program output provides values of V_u, V_c, and Av/s for selected segment locations of each span. Segment lengths are chosen not to exceed the beam section depth. The beginning of first segment and the end of last segment correspond to the locations of critical sections on the left and right support respectively. The critical sections are located at a distance “d”, the effective beam depth, away from the column face at both the left and the right ends of the beam. However, if concentrated loads are present within distance “d” from the column face, critical section is selected at the column face.

V_u is computed from the load acting over the entire width of the design strip. The program makes no distinction between shallow beams ($\alpha_1 l_2/l_1$ less than 1) and deeper beams ($\alpha_1 l_2/l_1$ greater than 1).

For ACI 318 Code the shear strength provided by concrete, V_c, is computed by:[40]

$$V_c = 2\sqrt{f_c^{'}}b_w d \qquad \text{Eq. 2-43}$$

In CSA design, for beams without minimum stirrup reinforcement and greater than 300mm deep, v_c is calculated from the following equation:

$$V_c = \left(\frac{260}{1000+d}\right)\lambda\varphi_c\sqrt{f_c^{'}}b_w d \geq 0.10\lambda\varphi_c\sqrt{f_c^{'}}b_w d \qquad \text{Eq. 2-44}$$

When $V_u > \varphi V_c / 2$, the beam must be provided with at least a minimum shear reinforcement of:[41]

$$\frac{A_v}{s} = \frac{50b_w}{f_{yv}} \qquad \text{Eq. 2-45}$$

Where

A_v = area of two legs of stirrups

s = stirrups spacing

b_w = longitudinal beam width

f_{yv} = yield strength of the shear reinforcement

When $V_u > \varphi V_c$, shear reinforcement must be provided so that:

$$\frac{A_v}{s} = \frac{V_u - \varphi V_c}{\varphi f_{yv} d} \qquad \text{Eq. 2-46}$$

Where

---

[40] ACI 318-99, 11.3.1.1; ACI 318-02, 11.3.1.1; CSA-A23.3-94, 11.3.5.1

[41] ACI 318-99, 11.5.5.2; ACI 318-02, 11.5.5.2; CSA-A23.3-94, 11.2.8.4

V_u = factored shear force at the section being considered

d = effective depth of the beam at the same location

For the ACI 318-99 Code the strength reduction factor for shear calculations is specified as φ=0.85. For the ACI 318-02 Code the strength reduction factor for shear calculations is equal φ=0.75.

When V_u exceeds $\varphi 5V_c$, the beam section dimensions must be increased or a higher concrete strength must be provided.[42] When $V_u \le \varphi 10\sqrt{f_c^{'}}b_w d$, the spacing is computed as:

$$s = \frac{1}{\frac{A_v}{s}} A_{sb} \qquad \text{Eq. 2-47}$$

Where

A_{sb} = total bar area of the two legged stirrup.

The maximum stirrup spacing must not exceed d/2 nor 24 in. when $V_u \le \varphi 6\sqrt{f_c^{'}}b_w d$. When $V_u > \varphi 6\sqrt{f_c^{'}}b_w d$, the maximum stirrup spacing must be reduced by half, to d/4 or 12 in.[43] When $V_u > \varphi 10\sqrt{f_c^{'}}b_w d$, the beam section dimensions must be increased or a higher concrete strength must be provided.

When no ribs are present (waffle) the program proportions the slab and beam share of one-way shear according to the following ratios:

$$\alpha_1 l_2 / l_1, \quad 1 - \alpha_1 l_2 / l_1 \qquad \text{Eq. 2-48}$$

When ribs are present (waffle) the program proportions the slab and beam share of one-way shear according to the following ratios of cross-section areas:

---

[42] ACI 318-99, 11.5.6.9; ACI 318-02, 11.5.6.9; CSA-A23.3-94, 11.3.4

[43] ACI 318-99, 11.5.4; ACI 318-02, 11.5.4; CSA-A23.3-94, 11.2.11

$$\frac{A_{ribs}}{A_{ribs} + A_{beam}}, \quad \frac{A_{beam}}{A_{ribs} + A_{beam}} \qquad \text{Eq. 2-49}$$

Area of Reinforcement

The program calculates the required area of reinforcement (top and bottom) based on the values of bending moment envelope within the clear span. For rectangular sections with no compression reinforcement, the design flexural strength of the column strip, middle strip and beam must equal the factored design moment:

$$M_u = \varphi f_y A_s \left(d - \frac{A_s f_y}{2(0.85 f_c^{'})b} \right) \qquad \text{Eq. 2-50}$$

The reinforcement can therefore be computed from:

$$A_s = \frac{0.85 f_c^{'} b}{f_y} \left(d - \sqrt{d^2 - \frac{2M_u}{\varphi 0.85 f_c^{'} b}} \right) \qquad \text{Eq. 2-51}$$

For CSA A23.3-94

$$M_r = \phi_s f_y A_s \left(d - \frac{\phi_s A_s f_y}{2(\alpha_1 \varphi_c f_c^{'} b} \right) \qquad \text{Eq. 2-52}$$

The effective depth of the section is taken as the overall section depth minus the distance from the extreme tension fiber to the tension reinforcement centroid. The column strip depth may include all or part of the drop panel depth. The drop depth will not be included in the effective depth of the column strip when the drop does not extend at least one-sixth the center-to-center span length in all directions, or when the drop depth below the slab is less than one-quarter the slab depth. If the drop does extend at least one-sixth the center-to-center span length and the drop depth is greater than one-quarter the distance from the edge of the drop panel to the face of the column or column capital, the excess depth will not be included in the column strip effective depth. If the drop width is less than the column strip width, the drop width will be used in the computation of the required reinforcement.

For the ACI 318-99 Code the strength reduction factor for flexure calculations is specified as ϕ=0.90.[44] For the ACI 318-02 Code the strength reduction factor for tension-controlled sections ($\varepsilon_t \geq 0.005$) is equal ϕ=0.90. For transition sections ($0.002 < \varepsilon_t < 0.005$) the strength reduction factor is specified by the formula:

[44] ACI 318-99, 9.3.2

$$\phi = 0.48 + 83 \cdot \varepsilon_t \qquad \text{Eq. 2-53}$$

ACI 318-02 Code specifies the strength reduction factor for compression controlled sections ($\varepsilon_t < 0.002$) as equal ϕ=0.65. The reduction factors for transition or compression controlled sections have application primarily in investigation mode of the program. In design mode the program performs the calculations assuming tension controlled section or section with compressive reinforcement (if enabled).

The ACI 318-99 Code requires keeping the steel ratio below 75 percent ρ_b, where:[45]

$$\rho_b = 0.85\beta_1 \frac{f_c^{'}}{f_y} \frac{87}{87 + f_y} \qquad \text{Eq. 2-54}$$

Where

β_1 = 0.85 for $f_c^{'} \le 4\text{ksi}$

= 0.65 for $f_c^{'} \ge 8\text{ksi}$

= $1.05 - 0.05 f_c^{'}$ for $4\text{ksi} < f_c^{'} < 8\text{ksi}$

For CSA code the value of ρ_b is calculated as follows:

$$\rho_b = \alpha_1 \beta_1 \frac{f_c^{'}}{f_y} \frac{700}{700 + f_y} \qquad \text{Eq. 2-55}$$

Where

$$\alpha_1 = 0.85 - 0.0015\, f_c^{'} \ge 0.67$$

$$\beta_1 = 0.97 - 0.0025\, f_c^{'} \ge 0.67$$

The ACI 318-02 Code controls the amount of reinforcement by limiting the value of net tensile strain ($\varepsilon_t \ge 0.005$) [46] (tensioned controlled section). From this condition the equivalent maximum reinforcement ratio for rectangular section can be written as:

---

[45] ACI 318-99, 8.4.3; CSA A23.3-94, 10.5.2

[46] ACI 318-02, 10.3.5

$$\rho_{max} = \frac{0.003}{0.003 + 0.005} \frac{0.85\beta_1 f'_c}{f_y} \qquad \text{Eq. 2-56}$$

If the calculated reinforcement exceeds the maximum allowed, a message will appear in the output. In such cases, it is recommended that the engineer review the slab thickness to ensure a more satisfactory design. If compression reinforcement calculations are enabled, **pcaSlab** will attempt to add compression reinforcement to the section. The program is capable to design compressive reinforcement for any design strip (column, middle, and beam) including also unbalanced moment strip.

The amount of reinforcement provided will not be less than the code prescribed minimum. For the ACI 318 Code, the minimum ratio of reinforcement area to the gross sectional area of the slab strip using Grade 60 reinforcement is taken as 0.0018. When reinforcement yield strength exceeds 60 ksi, the minimum ratio is set to 0.0018 x $60/f_y$. For reinforcement with yield strength less than 60 ksi, the minimum ratio is set to 0.0020. In no case will this ratio be less than 0.0014 [47] (See Table 2-6). The CSA Standard requires a minimum ratio of slab reinforcement area to gross sectional area of the slab strip equal to 0.002 for all grades of reinforcement.

f_y (ksi)	A_s/A_g
< 60	0.0020
≥ 60	$\dfrac{0.0018 \times 60}{f_y} \geq 0.0014$

Table 2-6 - Minimum Ratios of Reinforcement to Gross Concrete Area

According to ACI code for beams and positive moment regions of waffle slabs, minimum reinforcement provided will not be less than:[48]

$$A_{s,min} = \frac{3\sqrt{f'_c}}{f_y} b_w d \qquad \text{Eq. 2-57}$$

and not less than $200 b_w d/f_y$

Similar equation prescribed by CSA code has the form:

---

[47] ACI 318-99, 7.12.2.1; ACI 318-02, 7.12.2.1; CSA A23.3-94, 7.8

[48] ACI 318-99, 10.5.1; ACI 318-02, 10.5.1

$$A_{s,min} = \frac{0.2\sqrt{f_c^{'}}}{f_y} b_w h \qquad \text{Eq. 2-58}$$

In calculating required amounts of negative support steel for CSA A23.3-94 Code, the program performs the adjustment for banded and distributed reinforcement. The code requires concentrating one-third of the negative reinforcement at interior supports in the band region extending $1.5h_s$ from the sides of the columns. At exterior supports the total negative reinforcement is placed in the band region. The remaining portion of the negative reinforcement needs to be placed in the distributed strip. The reinforcement in banded and distributed strip is also checked for compliance with the minimum reinforcement requirement.

Reinforcement Selection

The default minimum clear spacing of reinforcement considered in **pcaSlab** is the Code prescribed minima for both slabs and beams of one bar diameter, d_b, or 1 in.[49] The user may select clear spacing greater than the default value.

For ACI 318 the maximum spacing of reinforcement is kept at two times the slab thickness.[50]

When calculating negative support reinforcement for CSA A23.3-94 the program assumes that banded reinforcement is spaced at a maximum of 1.5 h_s while distributed reinforcement is spaced at a maximum of 3 h_s.

An iterative process is performed to determine the number of bars and bar size. The initial number of bars is determined by dividing the total reinforcement area required, A_s, by the area of one bar, A_{sb}, of the input minimum bar size. Next, the spacing is determined. If the minimum spacing limitations are violated, the bar size is increased and the iterative process continues until all bars sizes have been checked. If the maximum spacing limitations are not met, the number of bars required to satisfy these limitations is computed and the iteration process terminates.

For beams, layered reinforcement is provided if sufficient beam width is not available. **pcaSlab** assumes a 1.5 in. cover to stirrup for width calculations. **pcaSlab** also assumes that the longitudinal bar makes contact at the middle of the stirrup bend where the

---

[49] ACI 318-99, 7.6.1; ACI 318-02, 7.6.1

[50] ACI 318-99, 13.3.2; ACI 318-02, 13.3.2

minimum inside diameter of the bend is $4d_b$. Therefore, an additional width is added to the cover for longitudinal bars less than #14 (an SI N-45) in size (Figure 2-20 – Figure 2-21). This additional width due to the bend, w_{bend}, is equal to:

$$w_{bend} = 0.293\left(1.0 - \frac{d_b}{2}\right) \qquad \text{Eq. 2-59}$$

Where

d_b = bar diameter of the longitudinal bar.

The clear distance between layers is assumed 1.0 in. Hooks and bends are not included in bar length figures.

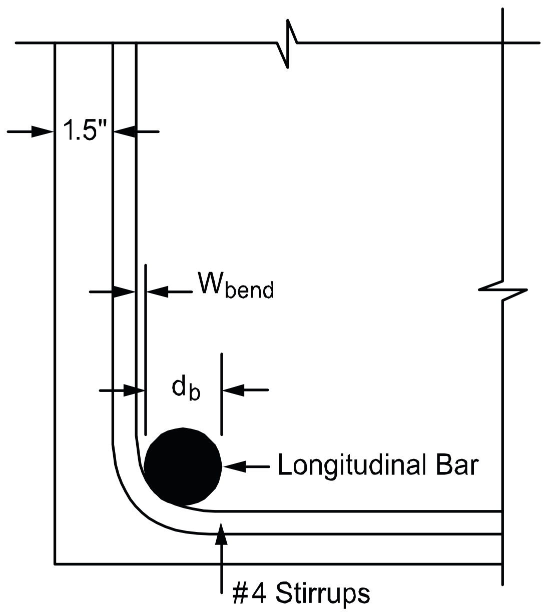


Figure 2-20 Width due to strip bend

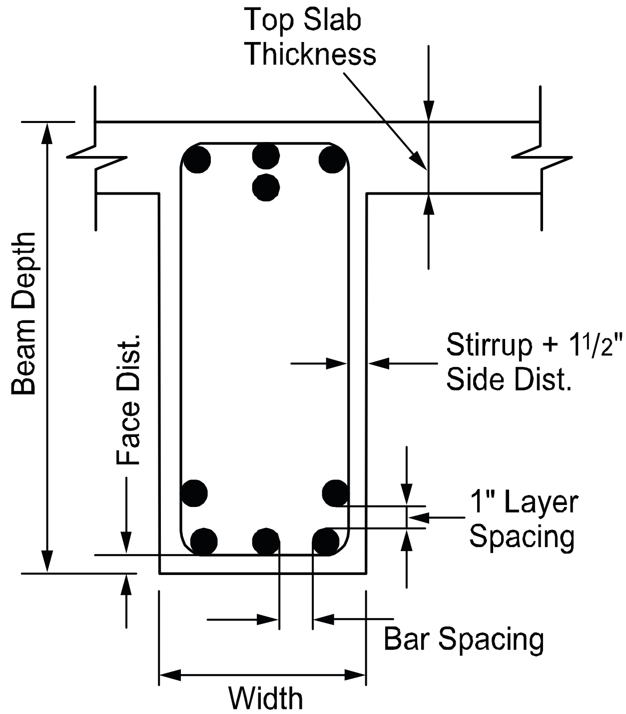


Figure 2-21 Detail reinforcement in longitudinal beams

Bar-length computations are performed for two-way slabs and longitudinal beams. For top reinforcement at the supports, the length for long bars is given by:

$$l_{long} = \max\begin{cases} \max(l_{50\%}) + l_{d,long} \\ \max(l_{pi}) + \max\begin{cases} d \\ 12d_b \\ l_n/16 \end{cases} \\ l_{fos} + l_{cr,long} \end{cases} \qquad \text{Eq. 2-60}$$

and the length for short bars is given by:

$$l_{short} = \max \begin{cases} \max(l_{50\%}) + \max \begin{cases} d \\ 12d_b \end{cases} \\ l_{fos} + \max \begin{cases} l_{d,short} \\ l_{cr,short} \end{cases} \end{cases} \qquad \text{Eq. 2-61}$$

Where

$\max(l_{50\%})$ = maximum distance to the points of 50% demand

$\max(l_{pi})$ = maximum distance to the points of inflection (P.I.)

l_d = bar development length[51]

d = effective depth

d_b = bar diameter

l_n = clear span length

l_{fos} = distance to the face of support (column)

l_{cr} = minimum code prescribed extension[46]

These bar lengths are then compared and adjusted if necessary to the minimum requirements specified by the code.[52] Additionally the program may select continuous top bars in those spans where steel is required by calculation in mid-span at top.

If the computed bar lengths overlap, it is recommended that such reinforcement be run continuously. The printed bar lengths do not include hooks or portions of bars bent down into spandrel beams or other bar-bend configurations.

The selection of bar lengths for positive reinforcement for flat plates, flat slabs, and beam-supported slabs, is based strictly on the minimum values of the Code. For waffle slabs, however, the positive reinforcement selection is based on an additional assumption of two bars per joist. For purposes of output, one bar is treated as a long bar and the other as a short bar.

The development length depends on the following factors: concrete cover, minimum transverse reinforcement, special transverse reinforcement, layer location bar size and

---

[51] ACI 318-99, Chapter 12; ACI 318-02, Chapter 12; CSA-A23.3-94, Clause 12.2

[52] ACI 318-99, Figure 13.3.8; ACI 318-02, Figure 13.3.8; CSA-A23.3-94, Figure 13.1

bar clear spacing. The computation of tension development lengths in terms of d_b is given by the general expression:

$$\frac{l_d}{d_b} = \frac{3}{40}\frac{f_y}{\sqrt{f_c'}}\frac{\alpha\beta\gamma\lambda}{\left(\frac{c + k_{tr}}{d_b}\right)} \qquad \text{Eq. 2-62}$$

Where

α = reinforcement location factor (1 or 1.3)

β = coating factor (1.2 or 1.5)

γ = reinforcement size factor (1 or 0.8)

λ = light weight aggregate concrete factor (1 or 1.3)

K_{tr} = transverse reinforcement index

c = spacing or cover dimension

Additional Reinforcement at Support

pcaSlab computes the fraction of the unbalanced moment, $\gamma_f M_u$, that must be transferred by flexure within an effective slab width equal to the column width plus one and one-half the slab or drop panel depth (1.5h) on either side of the column where:[53]

$$\gamma_f = \frac{1}{1 + \frac{2}{3}\sqrt{b_1 / b_2}} \qquad \text{Eq. 2-63}$$

The amount of reinforcement required to resist this moment is computed. The amount of reinforcement already provided for negative flexure is then computed from the bar schedule (i.e. the number of bars that fall within the effective slab width multiplied by the area of each bar). If the reinforcement area provided for flexure is greater than or equal to the reinforcement requirements to resist moment transfer by flexure, no additional reinforcement is provided, and the number of additional bars will be set to 0. If the amount of reinforcement provided for flexure is less than that required for moment transfer by flexure, additional reinforcement is required. The additional reinforcement is the difference between that required for unbalanced moment transfer by flexure and that provided for design bending moment in the slab, and it is selected based on the bar size already provided at the support.

[53] ACI 318-99, 13.5.3.2; ACI 318-02, 13.5.3.2; CSA-A23.3-94, 13.4.5.3

It should be noted that the Code requires either concentration of reinforcement over column beam joint by closer spacing, or additional reinforcement, to resist the transfer moment within the effective slab width. **pcaSlab** provides additional reinforcement, and does not concentrate existing reinforcement.

For the ACI Code, at least two of the column strip bottom bars should be continuous or spliced at the support with Class A splices or anchored with the support.

Integrity Reinforcement

For CSA A23.3-94 Code, the program performs calculation of the amount of integrity reinforcement at slab column connections. The integrity reinforcement is required for slabs without beams. The sum of all bottom reinforcement connecting the slab to the column on all faces of the periphery should meet the condition:

$$\sum A_{sb} \geq \frac{2V_{se}}{f_y} \quad \text{Eq. 2-64}$$

Corner Reinforcement

For CSA A23.3-94 Code, the program performs calculation of the amount of corner reinforcement in slabs with stiff edge beams (α greater than 1.0). This reinforcement is required within a region equal to 1/5 of the shorter span. The amount of corner reinforcement is calculated from the moment per unit width intensity corresponding to the maximum positive moment in span. The Code requires the corner reinforcement to be placed at top and bottom of the slab in bands parallel to the sides of the slab edges.

Deflection Calculation

Calculation of deflections of reinforced concrete two-way slabs is complicated by a large number of significant parameters such as: the aspect ratio of the panels, the vertical and torsional deflection of supporting beams, the stiffening effect of drop panels and column capitals, cracking, and the time-dependent nature of the material response. Based on studies (Ref. [22–24]), an approximate method consistent with the equivalent frame method was developed (Ref. [25]) to estimate the column and middle strip deflections.

Calculation of the midspan deflection of the column strip or the middle strip is based on the M/EI ratio of the strip to that of the full-width panel:

$$a_{f,strip} = a_{f,ref}\frac{M_{strip}}{M_{frame}}\frac{E_c I_{frame}}{E_c I_{strip}} \qquad \text{Eq. 2-65}$$

where $a_{f,ref}$ is taken from the equivalent frame analysis assuming gross section (uncracked) or effective (cracked) section properties.

The ratio (M_{strip}/M_{frame}) can be considered as a lateral distribution factor, LDF. The lateral distribution factor, LDF, at an exterior negative moment region is:

$$LDF_{neg,ext} = 100 - 10\beta_t + 12\beta_t\left(\alpha_1\frac{l_2}{l_1}\right)\left(1-\frac{l_2}{l_1}\right) \qquad \text{Eq. 2-66}$$

The LDF at an interior negative moment region is:

$$LDF_{neg,int} = 75 - 30\left(\alpha_1\frac{l_2}{l_1}\right)\left(1-\frac{l_2}{l_1}\right) \qquad \text{Eq. 2-67}$$

The LDF at a positive moment region is:

$$LDF_{pos} = 60 + 30\left(\alpha_1\frac{l_2}{l_1}\right)\left(1.5-\frac{l_2}{l_1}\right) \qquad \text{Eq. 2-68}$$

where

α_1 = the ratio of flexure stiffness of a beam section to the flexural stiffness of a width of slab bounded laterally by centerlines of adjacent panels on either side of the beam.

β_t = ratio of torsional stiffness of an edge beam section to the flexural stiffness of a width of slab equal to the span length of the beam, center-to-center of the supports (see Eq. 2-28).

When $\alpha_l l_2/l_1$ is greater than 1.0, $\alpha_l l_2/l_1$ will be set equal to 1.0.

The column and middle strip LDF's can be computed by:

$$LDF_c = \frac{LDF_{pos} + \frac{LDF_{neg,l} + LDF_{neg.r}}{2}}{2} \qquad \text{Eq. 2-69}$$

$$LDF_m = 100 - LDF_c \qquad \text{Eq. 2-70}$$

where

$LDF_{neg,l}$ = LDF for the negative moment region at the left end of the span

$LDF_{neg,r}$ = LDF for the negative moment region at the right end of the span

The deflections should be used in conjunction with the deflections obtained from an analysis in the transverse direction. For square panels ($l_1 = l_2$), the midpanel deflection is obtained by Eq. 2-71 as shown in Figure 2-22:

$$a = a_{cy} + a_{mx} = a_{cx} + a_{my} \qquad \text{Eq. 2-71}$$

For rectangular panels, ($l_1 \neq l_2$), the mid panel deflection is obtained by Eq. 2-72:

$$a = \frac{(a_{cy} + a_{mx}) + (a_{my} + a_{cx})}{2} \qquad \text{Eq. 2-72}$$

When calculating the deflections for effective (cracked) section properties, the equivalent frame solution is obtained for two load levels:

dead load only

dead load and live load combined

the effective section properties are assumed corresponding to the load level.

Cracking

If gross section properties option is selected for deflection calculations in the input, the deflection calculations will be based on the gross section I_g. If cracked section properties option is selected the effect of cracking on the deflection calculations is considered. The effect of cracking in slabs, as in beams, is a reduction in the flexural stiffness. The effect of cracking is generally an increase in deflections, although the tensile concrete between cracks continues to have a stiffening influence. A detailed review of several methods of deflection analyses can be found in Ref. [24].

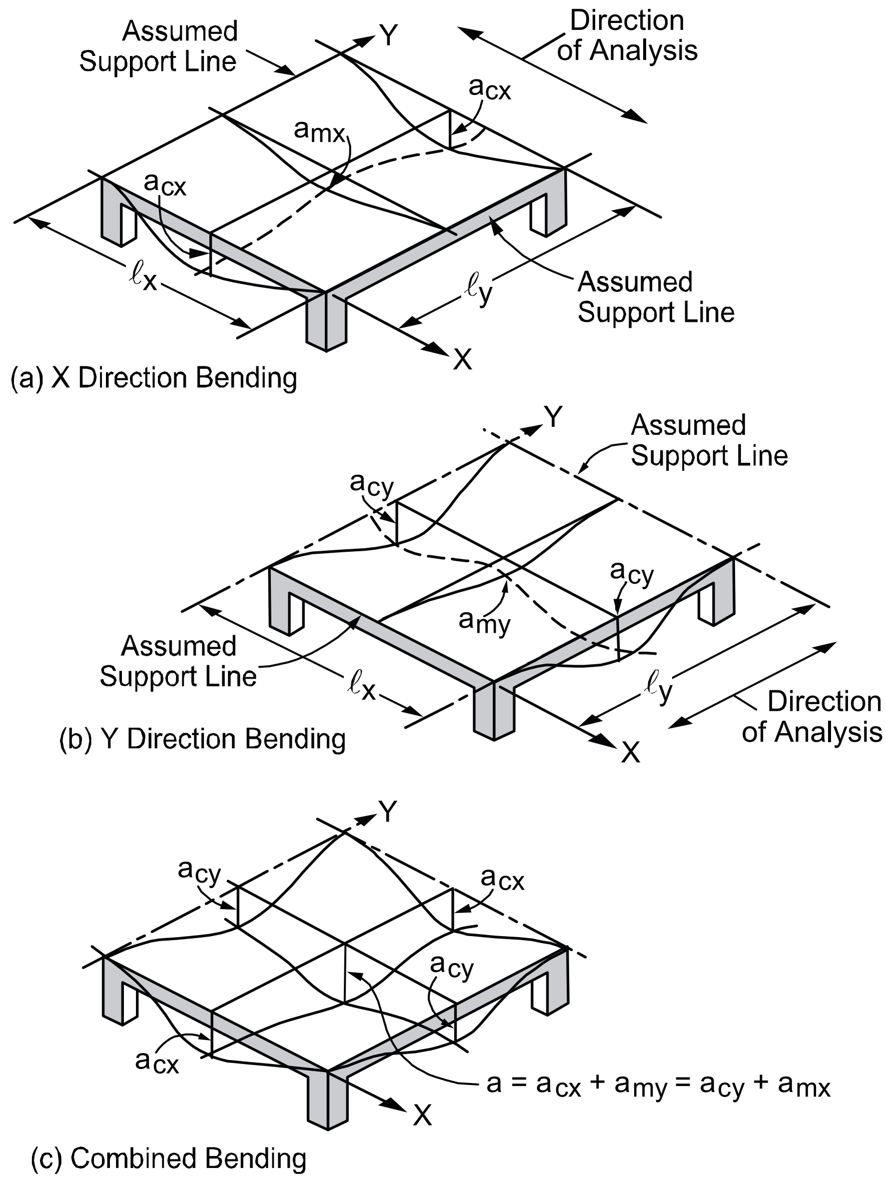


Figure 2-22 Deflection computation for a square panel

To apply the Code-specified effective moment of inertia to slab systems, a value of $I_{a,frame}$ is used instead. The value of $I_{a,frame}$ is given by:

$$I_{a,frame} = \beta I_e^+ + \frac{(1-\beta)\left(I_{e,l}^- + I_{e,r}^-\right)}{2} \qquad \text{Eq. 2-73}$$

Where

I_e^+ = effective moment of inertia for the positive moment region.

$I_{e,l}^-$ = effective moment of inertia for the negative moment region at the left support

$I_{e,r}^-$ = effective moment of inertia for the negative moment region at the right support

β = an empirical constant based on experimental data, whose value is less than 1.0

For flat plates, flat slabs, and waffle slabs, the appropriate value of β appears to be 0.5. **pcaSlab** uses a value of 0.5 for flat plates, flat slabs, and waffle slab systems and 0.7 for beam-supported slabs.

The value of $I_{a,frame}$ for cantilever spans is given by:

$$I_{a,frame} = \beta I_{e,sup}^- + (1-\beta) I_{e.end}^- \qquad \text{Eq. 2-74}$$

Where

$I_{e,end}^-$ = effective moment of inertia for the negative moment region at the cantilever end

$I_{e,sup}^-$ = effective moment of inertia for the negative moment region at support.

For flat plates, flat slabs, and waffle slabs, the appropriate value of β appears to be 1.0. **pcaSlab** uses a value of 1.0 for flat plates, flat slabs, waffle slabs, and beam-supported slabs.

The value of I_e corresponds to the effective moment of inertia as developed by Branson (Ref. [26]) and incorporated into the Code as follows:[54]

$$I_e = \left(\frac{M_{cr}}{M_{max}}\right)^3 I_g + \left[1 - \left(\frac{M_{cr}}{M_{max}}\right)^3\right] I_{cr} \qquad \text{Eq. 2-75}$$

Where

I_g = moment of inertia of the gross uncracked concrete section for the full width of the equivalent frame.

---

[54] ACI 318-99, 9.5.2.3; ACI 318-02, 9.5.2.3

I_{cr} = moment of inertia of the cracked transformed concrete section for the full width of the equivalent frame.

M_{cr} = cracking moment for the full width of the equivalent frame.

M_{max} = maximum bending moment for the full width of the equivalent frame at the load stage at which the deflection is computed.

It should be noted that the use of the full slab width in determining I_e in Eq. 2-75 is intended to average the effects of cracking of the column and middle strips.

Deflection Computations

Instantaneous Deflections: Two values of mid-span deflections are computed by the program for each span. The first corresponds to dead load on all spans, while the second corresponds to dead load plus live load on all spans. Note that the live load deflection is always computed as total load deflection minus the dead load defection. This is consistent with the I_{eff} method outlined in the Code.

Long Time Deflections: Long term deflection calculations are beyond the scope of the program. Users need to calculate long term deflection manually. The total mid-panel deflection (including long-time effects) consists of the following:

$$a_{total} = a_e + a_{creep} + a_{sh} \qquad \text{Eq. 2-76}$$

Where

a_e = elastic deflection

a_{creep} = additional deflection due to creap.

a_{sh} = additional deflection due to shrinkage

a_{total} = total deflection.

Data reported in the literature, although limited in quantity, indicate that the additional deflection due to creep and shrinkage, a_{creep} + a_{sh}, may be as much as two to eight times the elastic deflection, a_e (Ref. [27-30]). Such a factor times the dead load deflection may be used for an estimate of required slab camber.

For a crude approximation to a complex problem, additional long-time deflections may be computed as three times the elastic deflections. For a more detailed method of analysis of long-time deflection the user is referred to a study by Rangan (Ref. [30]).

Material Quantities

The program computes concrete and reinforcing steel quantities. The quantity of concrete is based on an average of the slab, drop, and beam sizes. The total quantity of reinforcing steel computed by the program corresponds to the actual bar sizes and lengths required by design. No allowance is made for bar hooks, anchorage embedment, and so forth. It should be noted that the quantity of reinforcement printed by the program pertains to bending in *one direction only*. In practice, the *total* amount of reinforcement for the structure should also include the quantities obtained for the appropriate transverse equivalent frames.

Ch. 3 User Interface Description

User Interface Components

Check Boxes

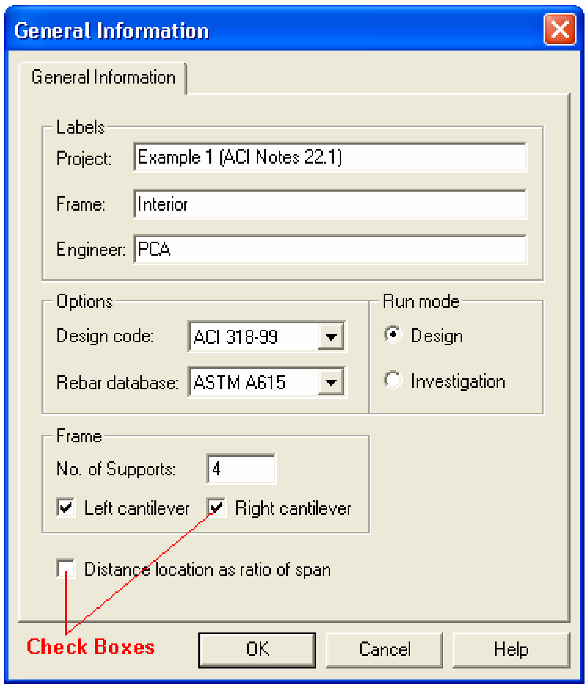


Figure 3-1 Check Boxes

A check box is a toggle used in a dialog box that enables or disables an option. To enable a check box, toggle the switch so that a ✓ sign is placed in the box. Removing the ✓ sign from the check box disables the command. Click once in the check box with the mouse to enable or disable the command or use the keyboard keys to tab to the check box and press the space bar to activate the option.

Checked Menu Commands

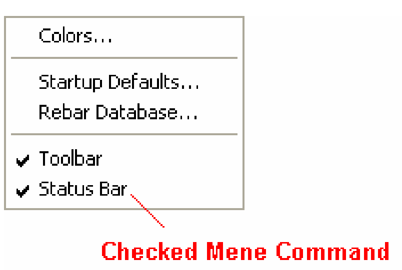


Figure 3-2 Checked Menu Commands

If the menu commands are checked with a ✓ sign on the left, the menu commands are enabled in the current project. To disable the command, single click the left mouse button on the command to clear the ✓ sign. For example, since the Toolbar and Status bar commands are checked in the image above, both the toolbar and status bar will appear in the main window of **pcaSlab**.

Command Buttons

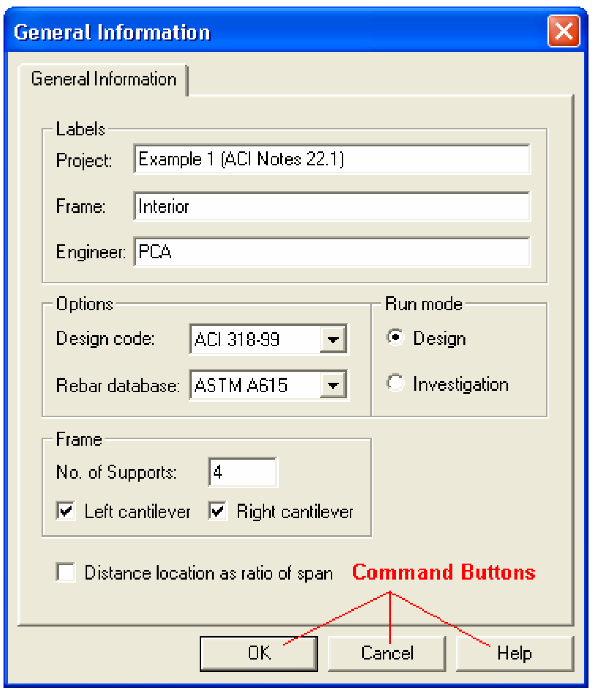


Figure 3-3 Command Buttons

Command buttons such as **Ok** and **Cancel** buttons that are located inside dialog boxes, are labeled to do exactly what they say. After you have completed your input within a dialog box, click once with the left mouse button on **Ok** (all Windows) or press **Alt** + **O** with the keyboard keys (Windows 95, 98, ME) to save the changes. If, while working in a dialog box, you want to disregard the data you have input, select **Cancel** (all Windows) to exit and discard any new data. When using the keyboard keys select **Alt** + **C** (Windows 95, 98, ME).

Control Menu

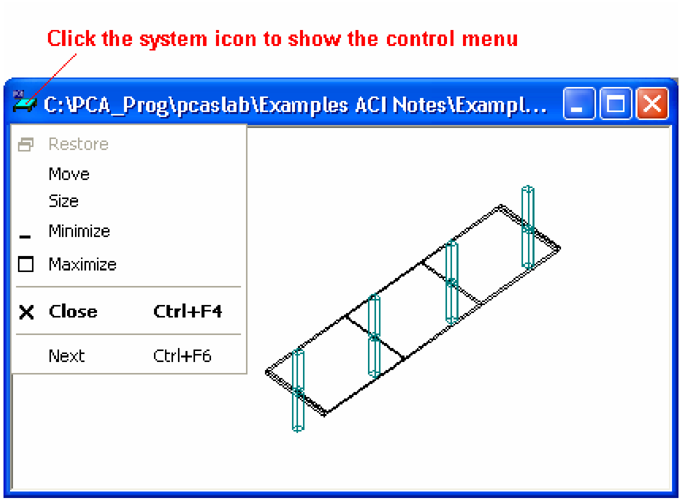


Figure 3-4 Control Menu

Each window has a Control menu located to the left of the title bar, in the upper left corner of the window. This menu contains commands that allow you to manipulate that particular window. To access the Control menu using the mouse, click the left mouse button on **pcaSlab** icon in the upper left corner. To access the Control menu using the keyboard, press **Ctrl** + **F6** to cycle through the windows until you reach the desired window and press **Alt + –** (hyphen).

Dialog Boxes

Dialog boxes are used to request information or provide you with information. By selecting a command from a menu, a dialog box may be displayed requesting data to complete the command. A dialog box will follow any command that contains three trailing periods, "**...**". After entering the required data, press the **Ok** button so that the data associated with that dialog box will be acted upon.

Drop-down List

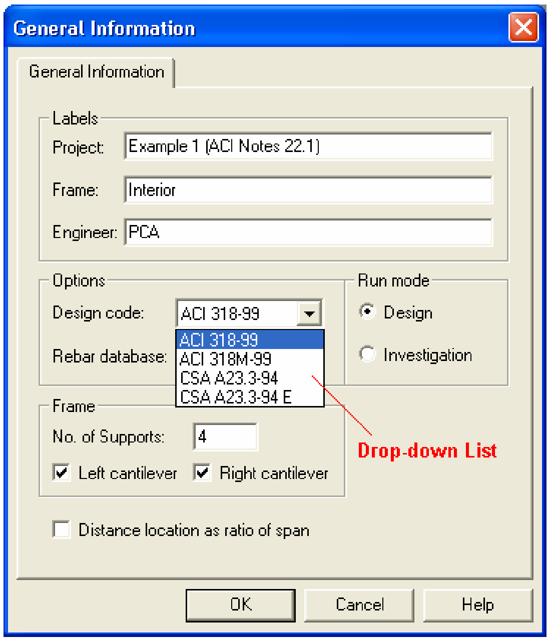


Figure 3-5 Drop-down List

A drop-down list is a list box that appears initially as a rectangular box with the current selection marked. To see a list of available choices, click the arrow to the right of the box or press **Alt + DOWN ARROW** to open the box. You can now select another item in a similar way as you would use a list box.

Usually there are some items in a drop-down list. You may click the left mouse button on the button on the right side of the list to extend the list. Then select the item you need by single clicking the left mouse button on the item. The selected item will appear in the text box on the top of the drop-down list. If there are too many items in a drop-down list, a vertical scroll bar will appear automatically so that you can use it to scroll up and down.

Drop-down Menu

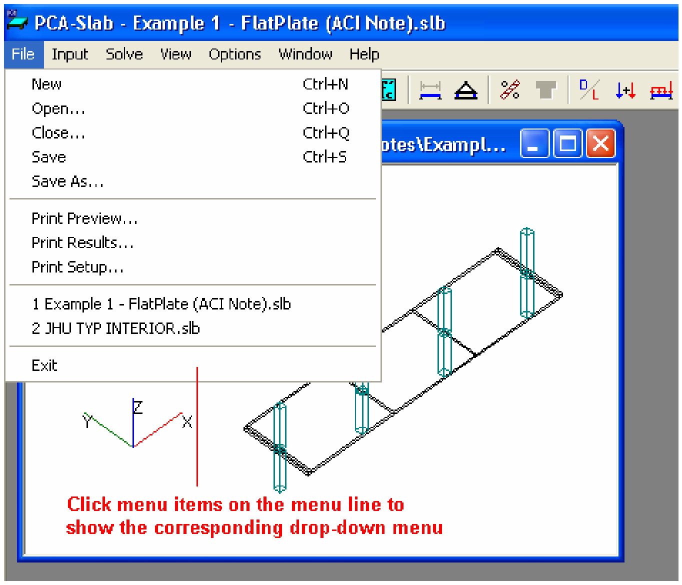


Figure 3-6 Drop-down Menu

A drop-down menu is the menu that is shown directly beneath the menu line when you click one of the menu items on the menu line using the left mouse button.

For example, the File menu appears directly beneath the "File" menu item when you click the "File" on the menu line or press **Alt + F** using keyboard.

Enable/Disable Options

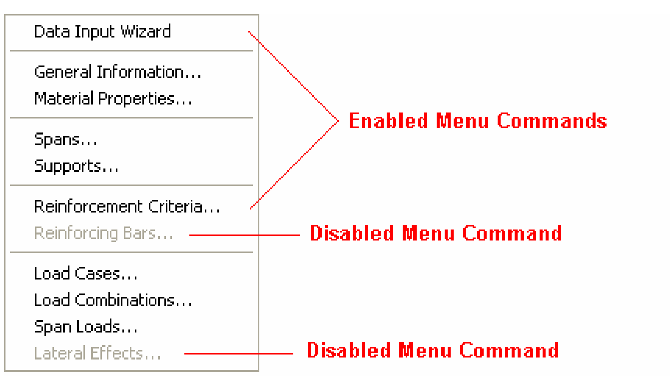


Figure 3-7 Enable/Disable Options

Some commands in the pop-up menus or input controls in the dialog boxes are black while others are gray. Commands that are black are available for use at that time. Commands that are grey are not available but may become available depending on the data you enter.

For example, in the **Input** menu, the **Reinforcing Bars** command is gray at the start of a new file. When you change the **Run Mode** from Design to Investigation in the **General Information** dialog box you will notice that the **Reinforcing Bar** command becomes black and is now ready for your input.

Frame Boxes

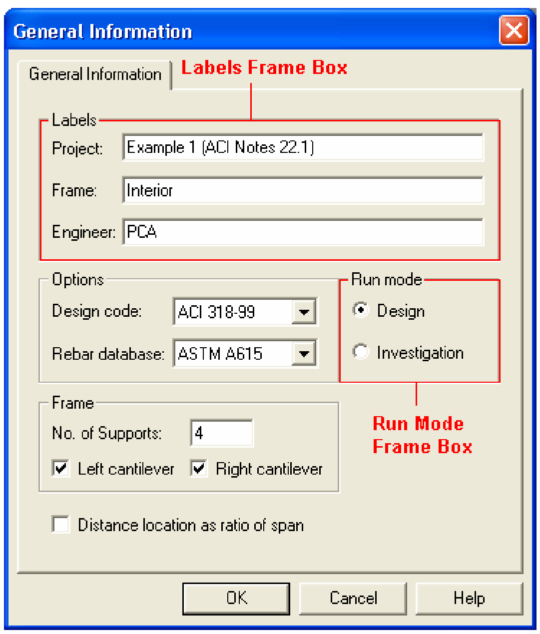


Figure 3-8 Frame Boxes

A frame box groups some related objects together, such as text boxes or option buttons.

List Boxes

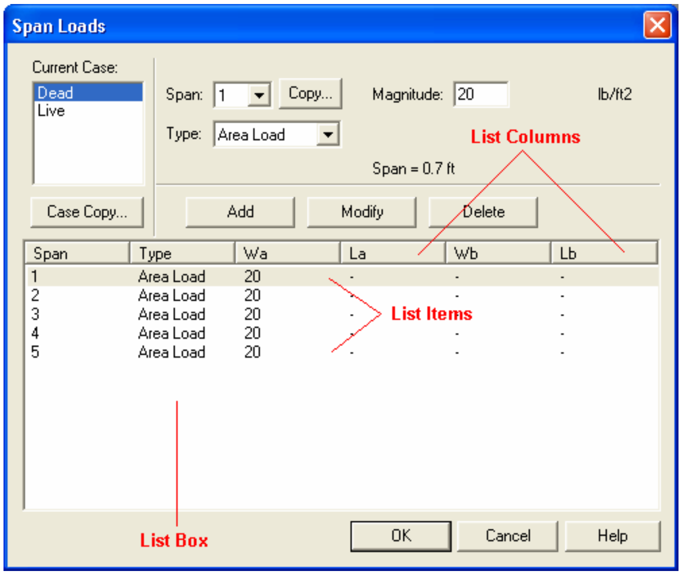


Figure 3-9 List Boxes

A list box displays a list of choices or items. If there are more choices than can fit in the box, scroll bars are provided so you can move quickly through the list.

The scroll bar is usually located to the right (vertical scroll bar) or at the bottom (horizontal scroll bar) of a list box. It has two directional arrows at either end (up and down, or left and right) in addition to a small rectangle called the scroll box. The location of the scroll box with respect to the length of the bar indicates the proportion of the amount of the information in view to the whole list.

To scroll through information displayed in a list box one line at a time, click the up or down scroll arrow or, using the keyboard, press the arrow key that points in the direction you want to scroll. Another way is to click inside the scroll bar above or below the scroll box. This scrolls the list one screen at a time (the keyboard equivalent is pressing the **Page Up** or **Page Down** key). If you drag the scroll box in the scroll bar, the section of the list that moves into view depends on where you position the scroll box.

To select a single item from a list box:

Click the scroll arrows until the item you want to select appears in the list box. Click the item.

Using the keyboard, use the directional arrow keys to scroll to the item you want.

To select multiple sequential items in a list:

Click the first item you want to select and then drag the cursor to the last item you want to select. Alternatively, click on the first item you want to select, press and hold down the **Shift** key, and click on the last item you want to select. Both items, and all items in between, are selected. To cancel the selections, release the **Shift** key and click any item.

Using the keyboard, use the **UP ARROW** or **DOWN ARROW** key to move the cursor to the first item you want to select. Press and hold down the **Shift** key. Continue to press the arrow key repeatedly until all the items you want are selected. To cancel the selections, release the **Shift** key and press one of the arrow keys.

To select multiple non-sequential items in a list:

Press and hold down the **Ctrl** key, and click each item you want to select. To cancel a selection, press and hold down the **Ctrl** key, and click the item again. There are no keyboard equivalents for this procedure.

Copying from a list box to the clipboard:

Selected items in a list box can be copied to the clipboard by simply pressing **F3** after you make your selection. You will be able to view the contents of the clipboard by opening the Clipboard Viewer in the Program Manager. Furthermore, the copied information can be transferred to other Windows applications (such as Microsoft Excel) using the application's Edit/Paste command. The transferred information will be indented and tabulated.

Option Buttons

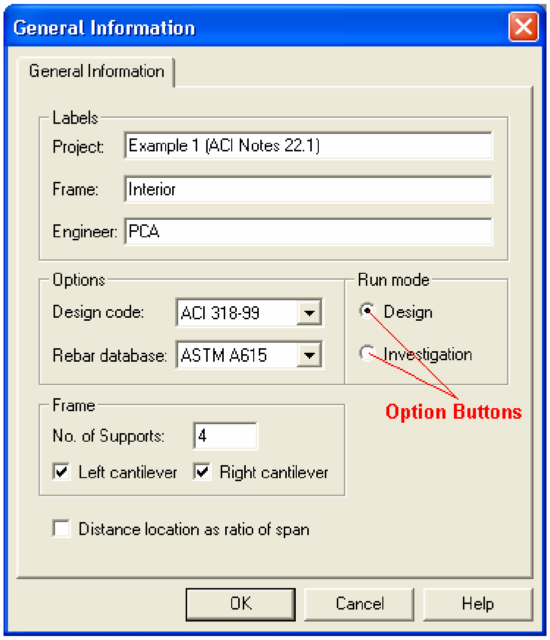


Figure 3-10 Option Buttons

An option button is used in a group of options from which you may select only one. A group of option buttons is a type of toggle (on/off) switch that when one option button is chosen, its state is toggled on and all other option buttons are toggled off. For example, in the General Information dialog box you will see option buttons. When using a mouse, place the cursor over the option desired and click the left button once to toggle the switch on. When using the keyboard keys, tab to the group of options and choose one by using the up and down arrow keys to toggle the switch.

Pop-up Menu

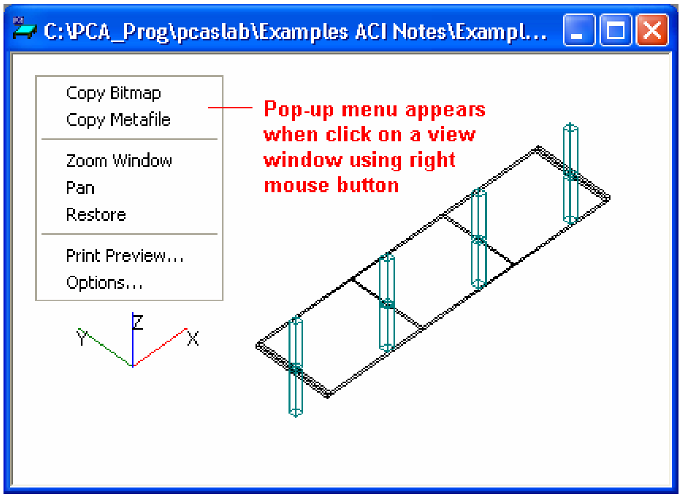


Figure 3-11 Pop-up Menu

A pop-up menu is the menu that is shown when you click the right mouse button on a view window. The commands on a pop-up menu are usually the commonly used commands associated with the view window that you right click on.

Tabs

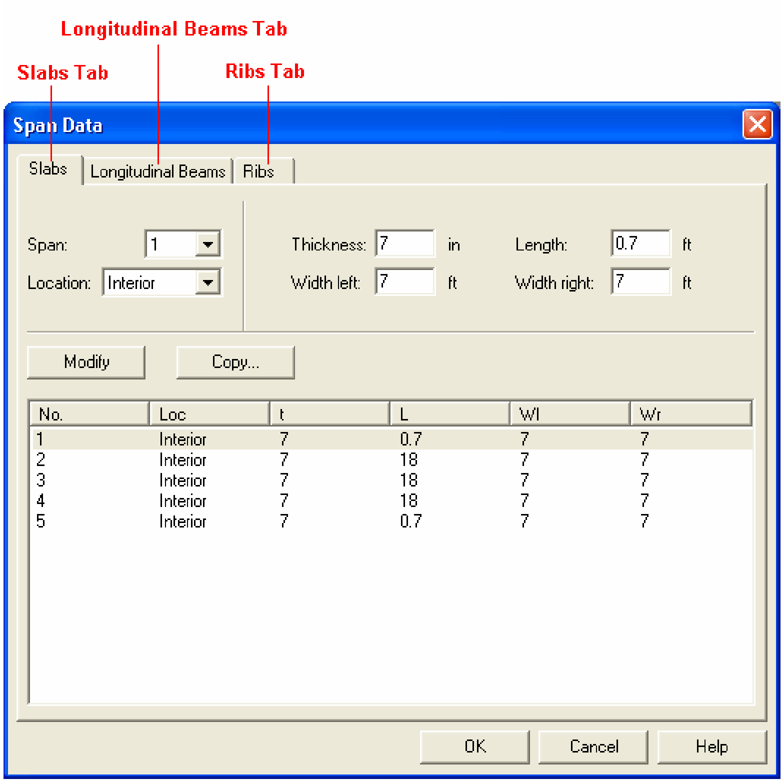


Figure 3-12 Tabs

Tabs are different dialog boxes that are integrated together in a more compact interface. Each tab is actually a dialog box itself and each tab has a tab title. For example, there are three tabs in the image shown above and the titles for them are Slabs, Longitudinal Beams, and Ribs, respectively. You may see different text boxes, option buttons, etc. on different tabs. To push a tab to the top or to activate a tab, single click the left mouse button on the title of the tab that you want to access. Once a tab is activated (on the top), other tabs are covered by the activated one and cannot be accessed.

Text Boxes

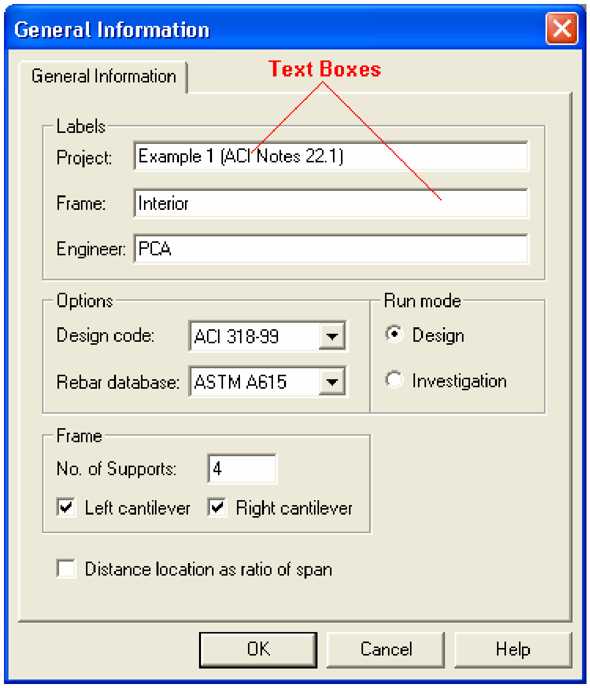


Figure 3-13 Text Boxes

Text Boxes are rectangular boxes inside dialog boxes that require text to be completed. In some cases, the edit box will contain only a flashing cursor located at the left side of the box prompting you to enter text. In other cases there may already be text in the text boxes. This text can be overwritten when you begin typing. To erase the text, use the backspace or the delete keys.

Main Window

Main Window

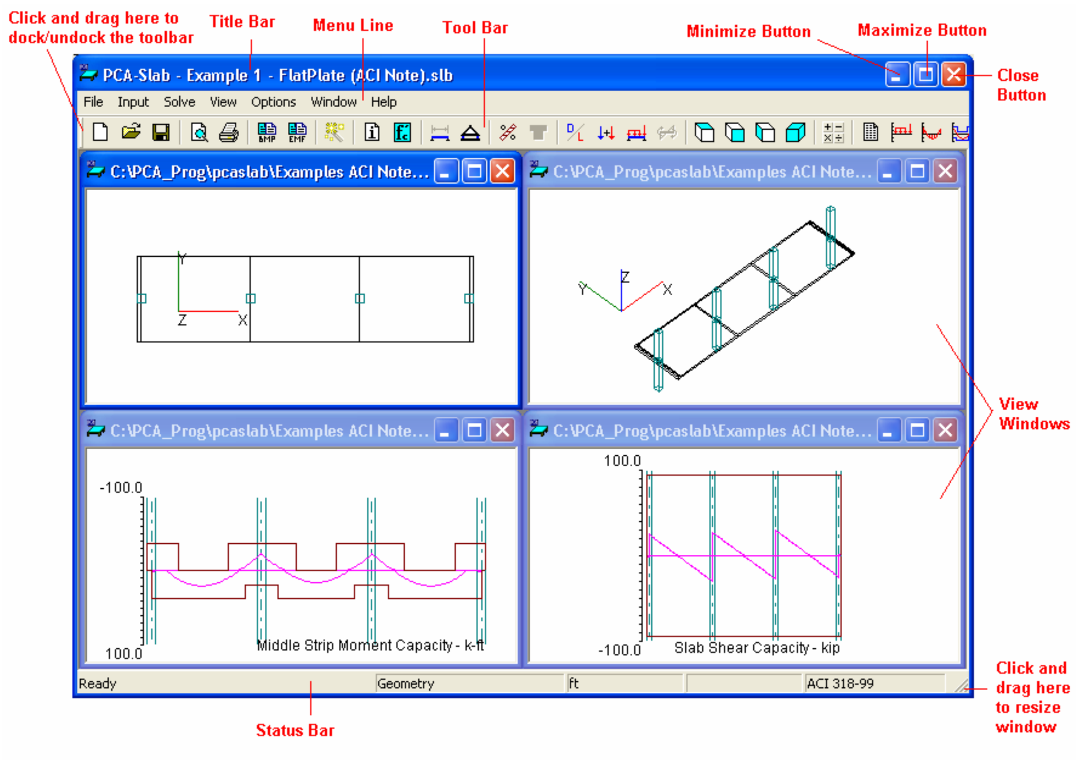


Figure 3-14 pcaSlab Main Screen

The main screen will appear after the **pcaSlab** is started as shown in Figure 3-14. The main screen consists of a title bar, menu line, tool bar, five view windows and status bar. The **pcaSlab** name and current data file name is shown in the title bar. All the menu commands can be accessed from the menu line and some frequently used commands also can be accessed from the buttons in the tool bar. The four view windows show the geometry of a floor system and the loads on it. Plan view, side view, elevated view and isometric view are available. The status bar shows the current states of **pcaSlab**.

Title Bar

Figure 3-15 Title Bar

The title bar displays the **pcaSlab** name, and following the hyphen, displays the name of the current data file you are using. If the data you are currently working on has not been saved into a file, the word *pcaSlab1* is displayed in the title bar. If you start an other new data file by clicking the **New** button on the most left of the tool bar, the next data file is named as *pcaSlab2*, and so on.

Menu Line

Figure 3-16 Menu line

Located directly below the title bar is the menu line. **pcaSlab** commands are listed in the pop-up menus located in the menu line. These menu commands allow you to perform functions that create, view, and ultimately design the floor system.

In the **pcaSlab** program there are seven main pop-up menus: **File**, **Input**, **Solve**, **View**, **Options**, **Windows**, and **Help**. To access a menu item using the mouse, place the arrow cursor on the menu item you want and click the left mouse button. Each menu item can also be selected with the keyboard keys by simultaneously pressing the **Alt** key and the underlined letter of the menu you want to open. For example, to open the File menu, press **Alt** + **F**. To close a menu without selecting a command, move the cursor to any blank area on the screen and click the left mouse button. Press **Esc** key to close a menu using the keyboard keys.

To select a command from a menu with the mouse, place the arrow on the item you want, and click the left mouse button. In some cases, you will be told to double click on a selection, that is, press the mouse button twice, quickly. Anytime you have to wait, for example, when loading the program or designing the system, the mouse cursor becomes an hourglass cursor. It will return to its original state when the task is completed.

To select a command from a menu using the keyboard, use the down arrow key to highlight your choice and press **Enter** key or press the keyboard key of the command's underlined letter. The space bar is also equivalent to pressing the left mouse button.

Special instructions for inputting with the keyboard keys are given wherever necessary.

Tool Bar

Figure 3-17 Tool Bar

Located directly below the menu line is the tool bar. Some frequently used buttons can be found in the tool bar. A description of the corresponding button is shown in the status bar (on the bottom of the window) when the mouse cursor is moving over this button. In addition to the description in the status bar, a brief tip is shown in a light yellow colored pop-up window close to the corresponding button when a mouse cursor is hanging over the button for a short period of time. Exactly the same functions or features can be accessed from either the menu items or tool bar buttons.

The tool bar can be changed from docking status to floating status by single clicking the left mouse button on the tool bar and dragging it away from the docking position to any other positions on the screen. A floating tool bar can be resized by clicking and dragging its borders.

To restore the tool bar to the docking status, single click the left mouse button on the tool bar and drag it to the location that is directly below the menu line and release the mouse button.

View Windows

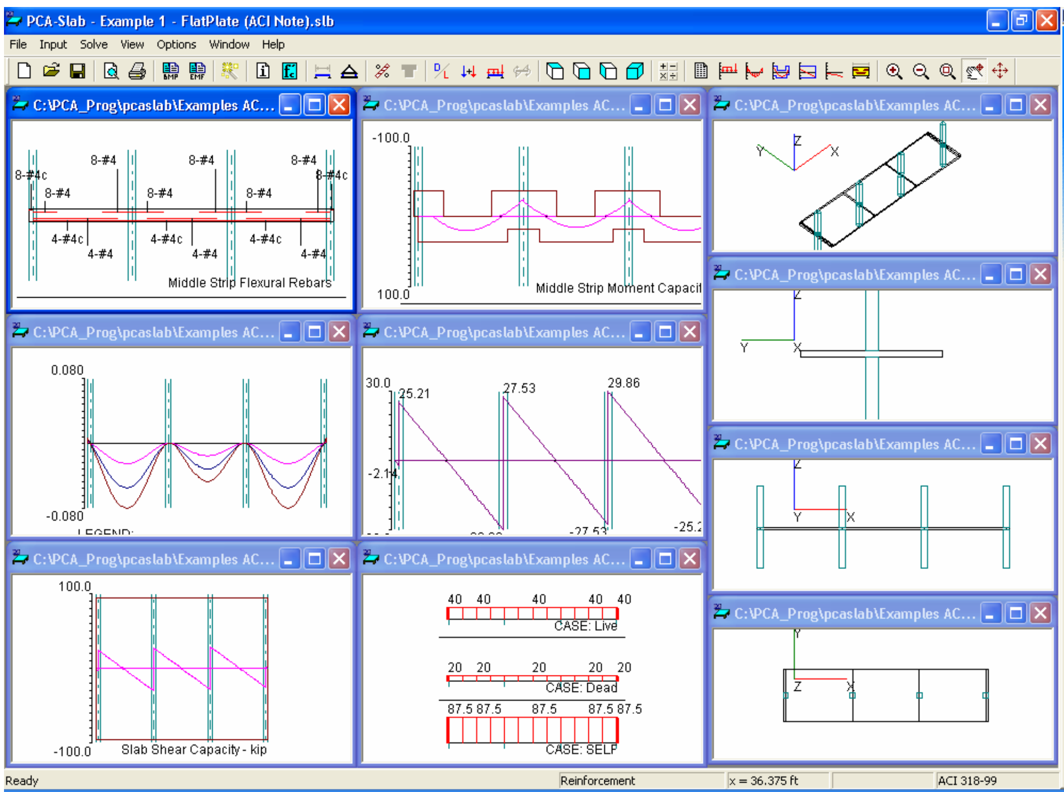


Figure 3-18 View Windows

A total of 10 view windows can be used to show Plan, Elevated, Side and Isometric views of the geometry, as well as Loads, Shear and Moment, Moment Capacity, Shear Capacity, Deflection, and Reinforcement.

Status Bar

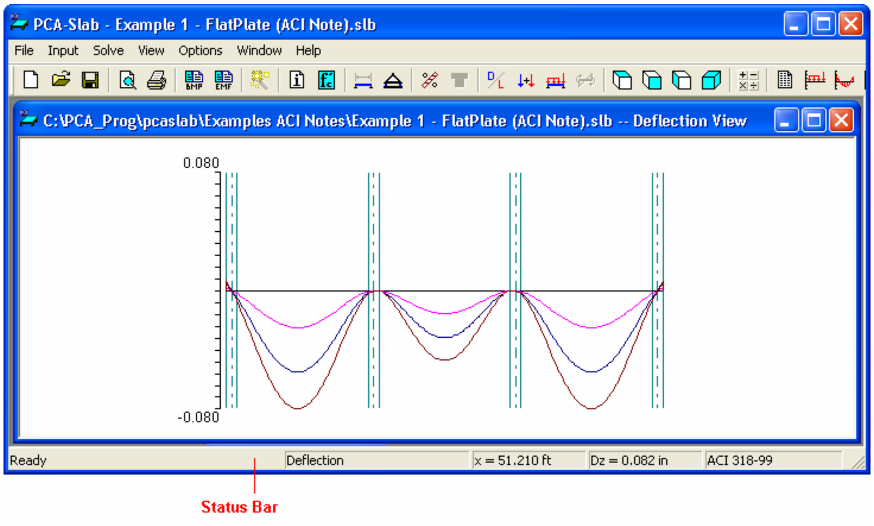


Figure 3-19 Status Bar

Status bar is always on the bottom of the main window. The status bar shows the current status of the data file and the coordinate values of the mouse cursor position.

For example, Figure 3-19 shows that **pcaSlab** is ready to take user input and the current view window is Deflection View. The "x=31.835 ft" is the horizontal coordinate of the mouse cursor (shown as "+" symbol) and the "D_z=-0,042 in" shows the current vertical value of the mouse cursor. Depending on the active view window, different information of the mouse cursor will appear in the status bar.

Main Menu

File Menu

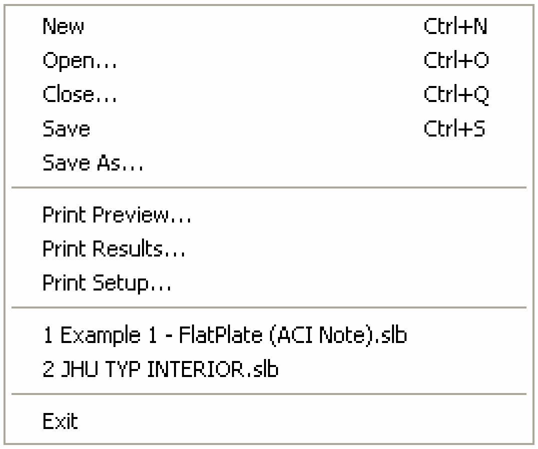


Figure 3-20 File Menu

The **File** menu is used for saving or retrieving data, printing, and exiting. The **File** menu contains the following commands: **New**, **Open**, **Close**, **Save**, **Save As**, **Print Preview**, **Print Results**, **Print Setup**, **Recent Files** and **Exit**.

New

The **New** command clears any data input and returns to the default values. Thus, you are able to create a new data file. However, before you can begin a new data file, **pcaSlab** will ask whether you want to save the current data as shown in Figure 3-21. Answering **Yes** will save the old data and begin a new data file. Answering **No** will discard any changes to the data and begin a new data file. Answering **Cancel** will return you to **pcaSlab** so that you can continue to work with the current data.

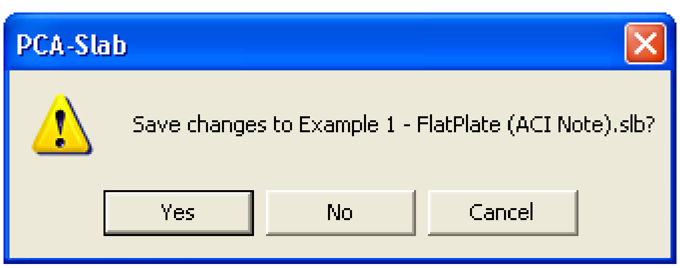


Figure 3-21 Save Change Dialog Box

Open

The **Open** command allows you to load an existing **pcaSlab** data file. The dialog box that appears shows you a listing of all the files with the extension contained in the default data directory or in the current directory (if a default data directory was not specified). This box also enables you to change the current drive and directory. If you are currently working on a data file and select the **Open** command, **pcaSlab** will ask whether you want to save the current data. Answering **Yes** will save the old data and display the Open dialog box. Answering **No** will discard any changes to the data and display the Open dialog box. Answering **Cancel** will return you to **pcaSlab** so that you can continue to work with the current data.

Close

The **Close** command allows you to close the current **pcaSlab** data file. If you are currently working on a data file and select the **Open** command, **pcaSlab** will ask whether you want to save the current data. Answering **Yes** will save the old data and display the Open dialog box. Answering **No** will discard any changes to the data and display the Open dialog box. Answering **Cancel** will return you to **pcaSlab** so that you can continue to work with the current data.

Save

The **Save** command saves the changes you've made to the current data under that same filename. The new data overwrites the old data, and you cannot retrieve the old data. It is a good practice to periodically save while inputting data. If a data file is untitled, the Save As dialog box will appear as shown in Figure 3-22.

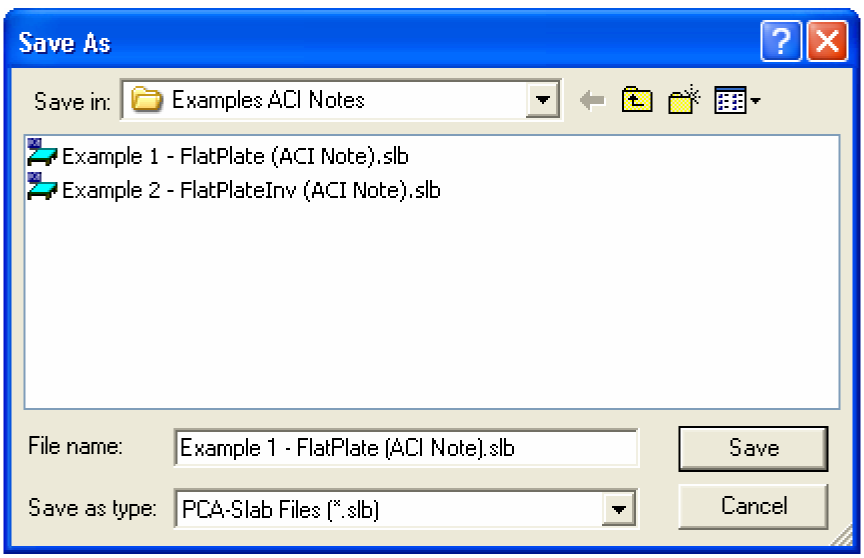


Figure 3-22 Save As Dialog Box

Save As

The **Save As** command allows you to name or rename a data file. Use **Save As** when you want to save both the original data and any changes you've currently made to the data. The original data remains under the old filename. If a file of the same name exists, the program will ask if you would like to overwrite the file.

Print Preview

The **Print Preview** command allows you to preview and print the current view window (floor system geometry in the plan, elevated, and isometric views, prints the shear and moment diagrams, and prints the deflected shapes). To obtain a view window you must first perform the design, then select what you want to view from the **View** menu. You may have more than one view windows opened. The current view window is the one activated and on top of the others on your screen. Selecting this command closes the **pcaSlab** main window and opens the print preview window as shown in Figure 3-23. On the print preview window, press the **Zoom In** or **Zoom Out** buttons or simply click the left mouse button on the preview window to magnify or reduce the size of the preview paper. Press the **Next Page** button if more than one page needs to be printed. Press the **Print** button to print the view. The printer could be a local printer, which is connected to your computer directly, or a network printer. Press the **Close** button to close the preview window and go back to **pcaSlab**.

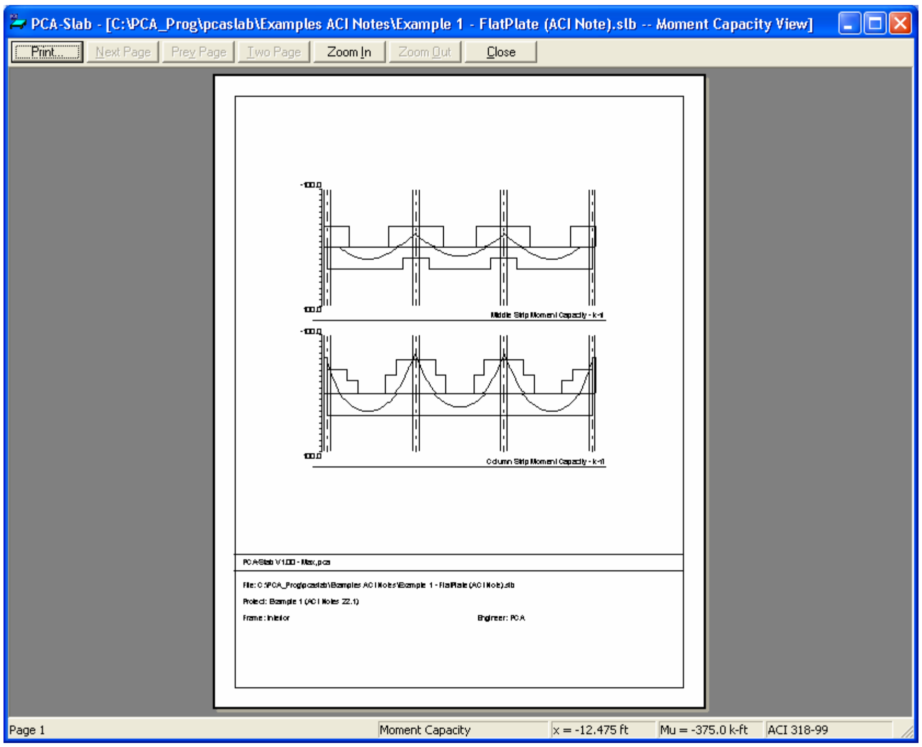


Figure 3-23 Print Preview Window

Print Results

The **Print Results** command allows you to send the analysis and/or design results to the printer. Selecting this command will show the Result Options dialog box as shown in Figure 3-24. By selecting options in the Output Options dialog box prior to performing the print, you can print selected sections of the data instead of the entire analysis.

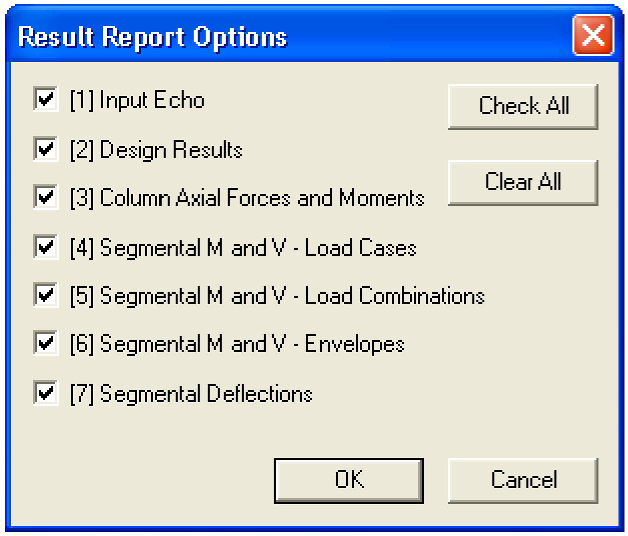


Figure 3-24 Result Options Dialog Box

Recent Files

This list contains the data files that are used recently and can be accessed quickly from the menu by a single click. The most recently used one is on the top of the list. Up to four files can be listed.

Exit

The **Exit** command ends the **pcaSlab** session and returns you to Windows. If you've made any changes to your data and have not saved them, **pcaSlab** will first ask whether you want to save or abandon any changes you've made before you exit.

Input Menu

Figure 3-25 Input Menu

The **Input** menu allows you to enter and modify data for the floor system. The **Input** menu contains the following commands: Data Input Wizard**,** General Information**,** Material Properties**,** Spans**,** Supports**,** Reinforcement Criteria**,** Reinforcing Bars**,** Load Cases**,** Load Combinations**,** Span Loads and Lateral Effects**.**

Data Input Wizard

The **Data Input Wizard** command is designed to make the inputting process easier. By selecting **Data Input Wizard**, a logical sequence of dialog boxes will automatically be displayed allowing you to enter data for your floor system.

General Information

The **General Information** command will allow you to enter the project name, frame name, engineer name, design code, rebar database, run mode, and number of supports. The General Information dialog box is shown as Figure 3-25. You must always choose the General Information command before doing any further inputting since it affects the availability of other commands in this menu.

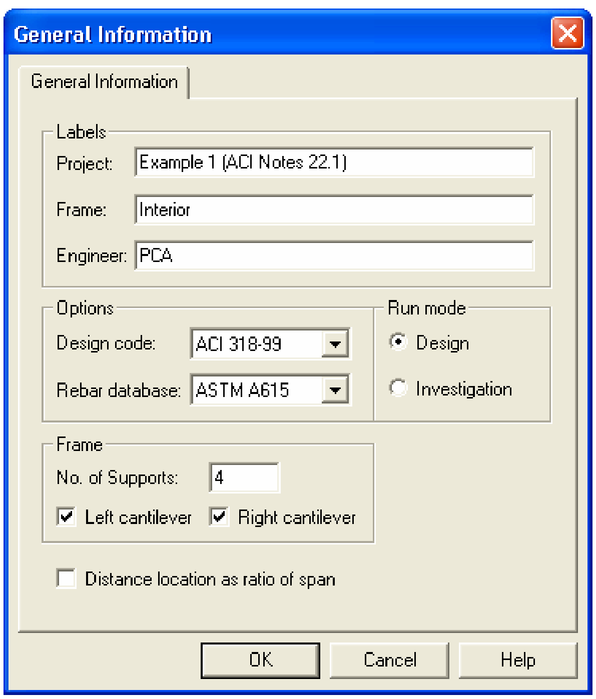


Figure 3-26 General Information Dialog Box

Material Properties

The **Material Properties** command enables you to input material property requirements for concrete and reinforcement, as shown in Figure 3-27, and Figure 3-28, respectively.

Concrete density, compressive strength, Young's modulus, rapture modulus, as well as the longitudinal and shear reinforcement yield levels, are required.

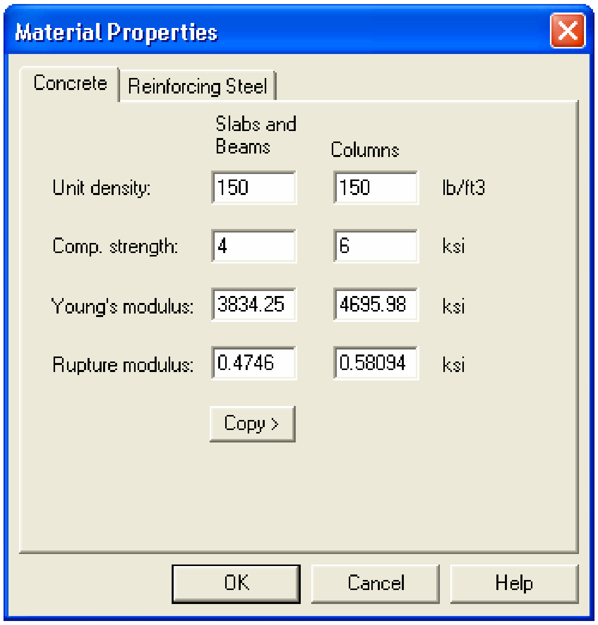


Figure 3-27 Concrete Properties Dialog Box

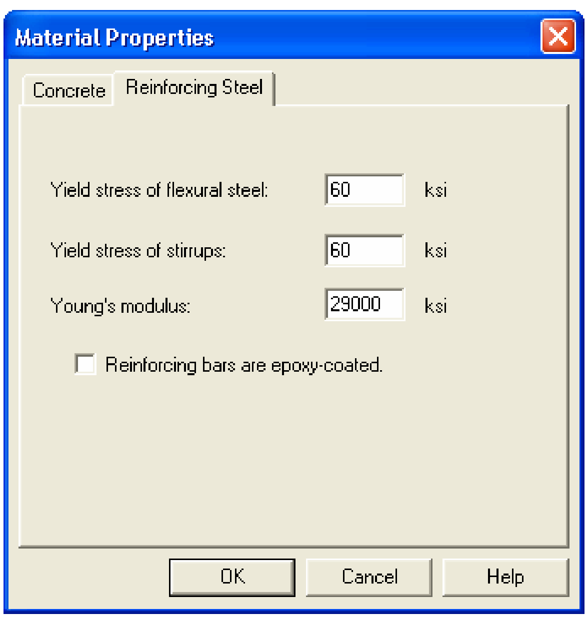


Figure 3-28 Reinforcing Steel Properties Dialog Box

Spans

The **Spans** menu allows you to input geometric dimensions for slabs, longitudinal beams, and ribs, as shown in Figure 3-29.

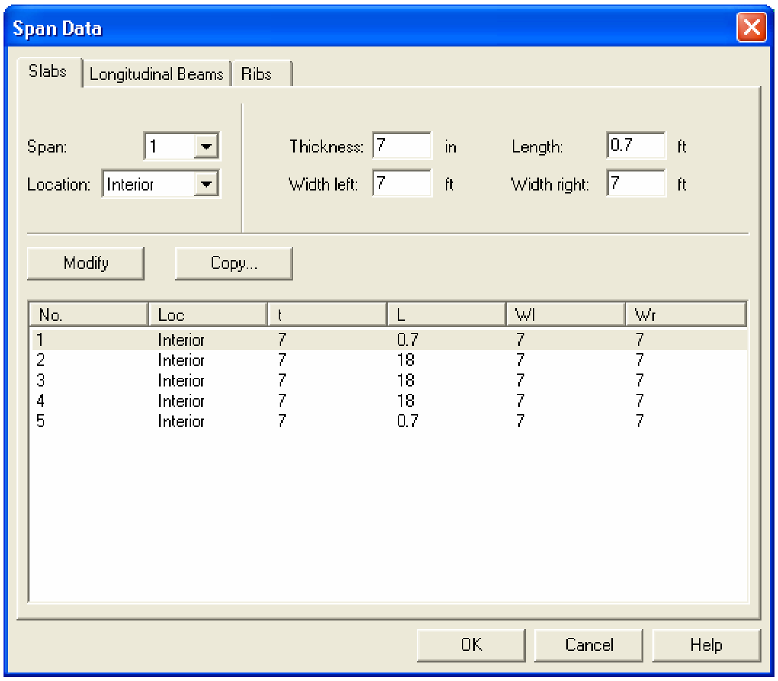


Figure 3-29 Spans Dialog Box

Supports

The **Supports** menu allows you to input geometric dimensions for columns, drop panels, column capitals, and transverse beams, as shown in Figure 3-30. The percentage of the actual column joint stiffness to be used in the analysis to determine the joint moments and shears can be modified on the **Columns** tab.

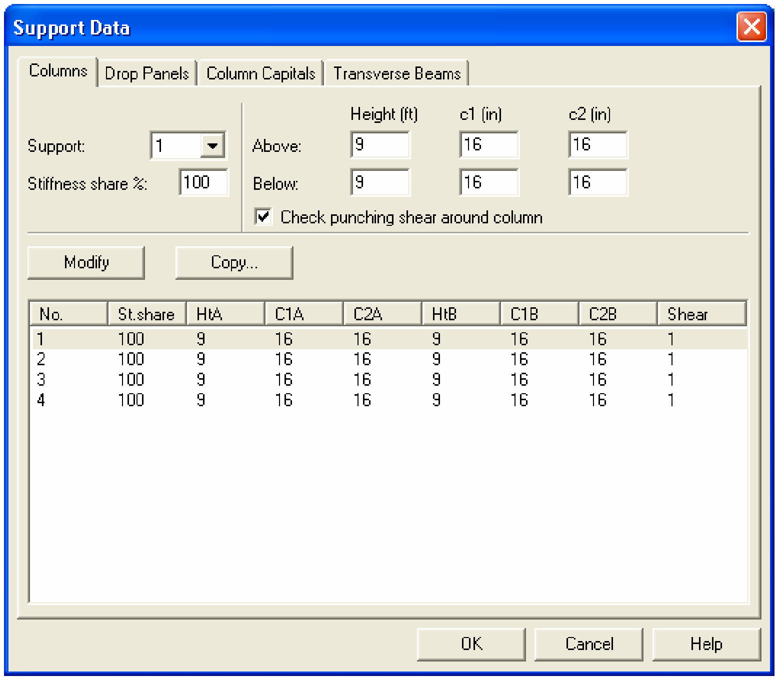


Figure 3-30 Supports Dialog Box

Reinforcement Criteria

The **Reinforcement Criteria** menu allows you to specify the distance to reinforcement, reinforcement bar sizes, bar spacing, and reinforcing ratio for both slabs and beams.

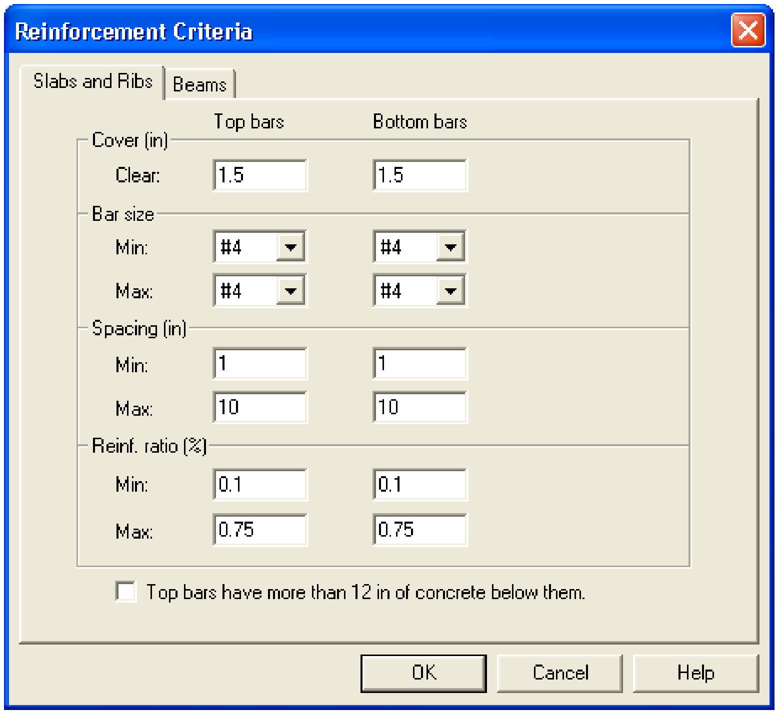


Figure 3-31 Reinforcing Criteria Dialog Box

Reinforcement Bars

The **Reinforcement Bars** menu allows you to specify the longitudinal reinforcement arrangement information for column strip, middle strip, and beam, as well as shear reinforcing information for beams. The **Reinforcement Bars** menu is disabled if Run Mode of Design is selected from the General Information dialog box. Select the Run Mode of Investigating from the General Information dialog box to enable it.

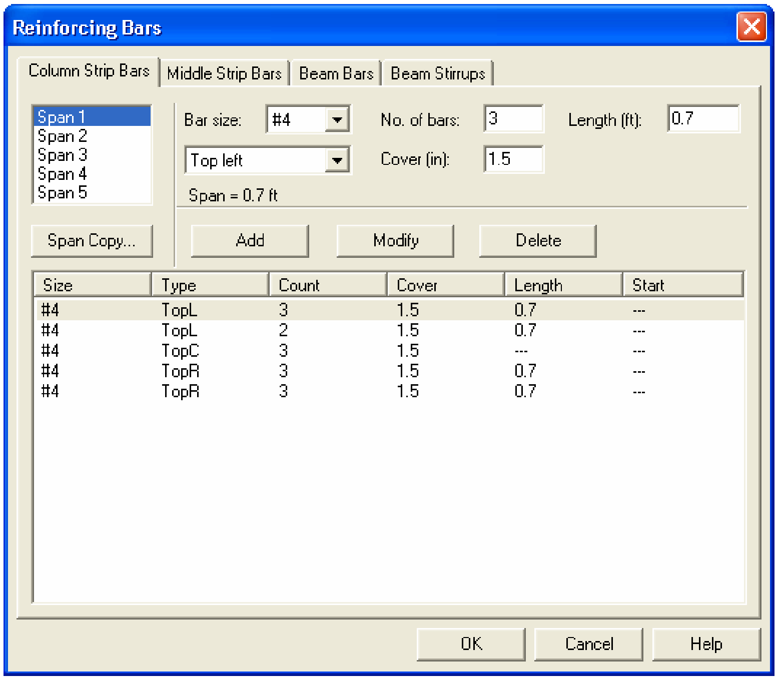


Figure 3-32 Reinforcing Bars Dialog Box

Load Cases

The **Load Cases** menu allows you to specify load cases as shown in Figure 3-33. Up to six load cases can be added and only one live load case is allowed.

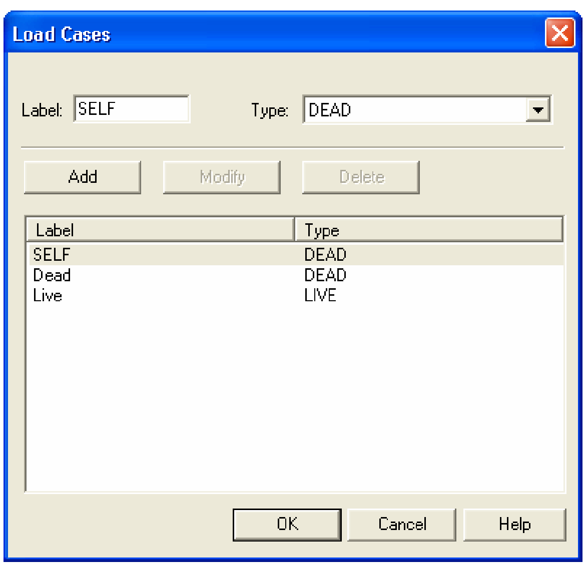
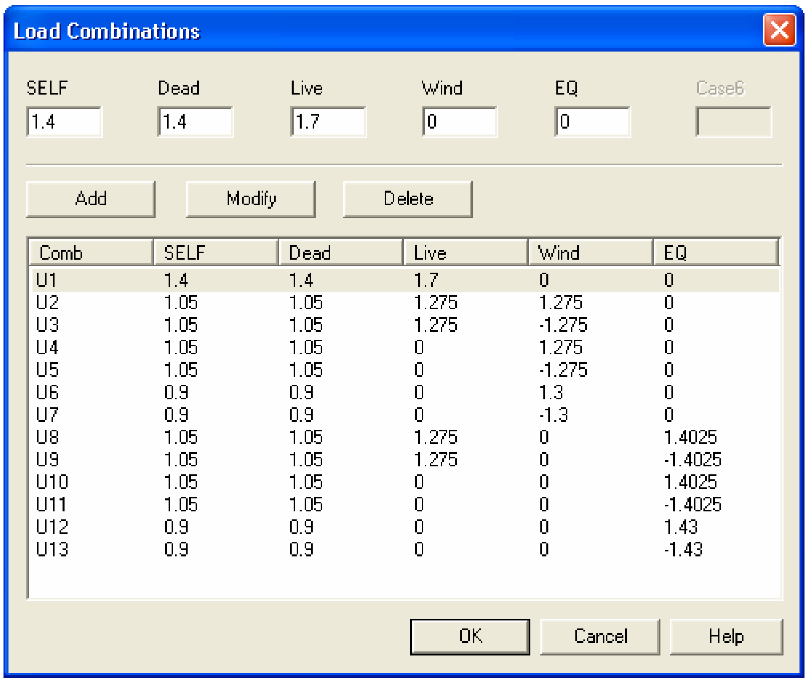


Figure 3-33 Load Cases Dialog Box

Load Combinations

The **Load Combinations** menu allows you to specify load combinations as shown in Figure 3-34. Up to twelve load combinations can be added.



Comb	SELF	Dead	Live	Wind	EQ
U1	1.4	1.4	1.7	0	0
U2	1.05	1.05	1.275	1.275	0
U3	1.05	1.05	1.275	-1.275	0
U4	1.05	1.05	0	1.275	0
U5	1.05	1.05	0	-1.275	0
U6	0.9	0.9	0	1.3	0
U7	0.9	0.9	0	-1.3	0
U8	1.05	1.05	1.275	0	1.4025
U9	1.05	1.05	1.275	0	-1.4025
U10	1.05	1.05	0	0	1.4025
U11	1.05	1.05	0	0	-1.4025
U12	0.9	0.9	0	0	1.43
U13	0.9	0.9	0	0	-1.43

Figure 3-34 Load Combinations Dialog Box

Span Loads

The **Span Loads** menu allows you to enter superimposed area loads, line loads, point loads, and moments.

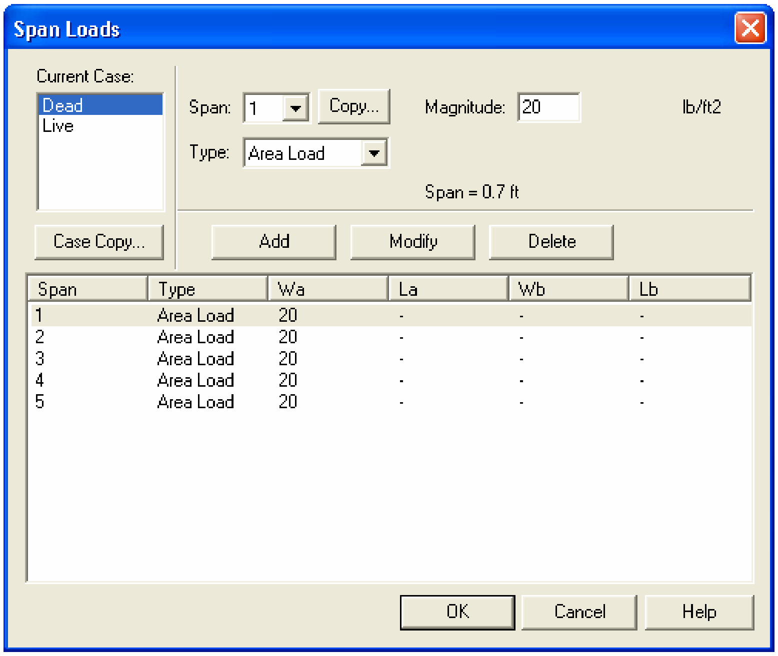


Figure 3-35 Span Loads Dialog Box

Lateral Effects

The **Lateral Effects** menu allows you to enter the lateral loads as moments acting on the two ends of each span.

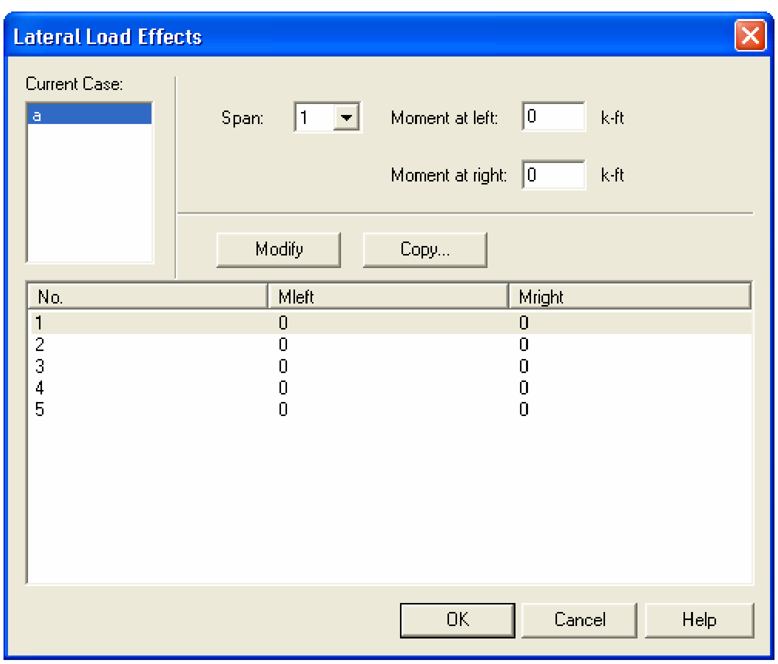


Figure 3-36 Lateral Effects Dialog Box

Solve Menu

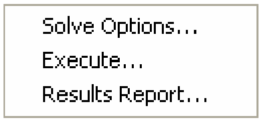


Figure 3-37 Solve Menu

The **Solve** menu contains commands that enable you to perform the analysis and/or design of the floor system and view the results. The **Solve** menu contains the following commands: **Solve Options**, **Execute**, and **Report**.

Solve Options

The **Solve Options** command allows you to specify the live load pattern ratio, critical section of punching shear, and the section to calculate deflection, as shown in Figure 3-38. To take effect, the design must be performed after the Condense/Expand Output command is executed.

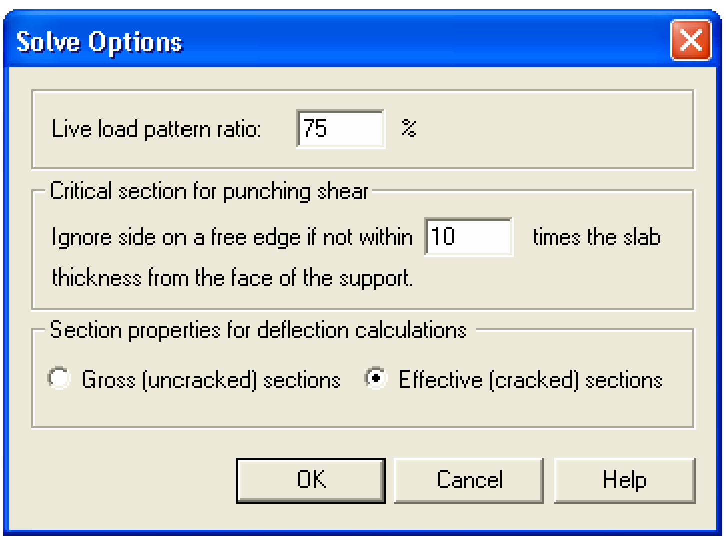


Figure 3-38 Solve Options Dialog Box

Execute

The **Execute** command executes the solver portion of **pcaSlab**. If some data is still required when this command is executed, **pcaSlab** will respond with an "Invalid Model!" error message, as shown in Figure 3-39.

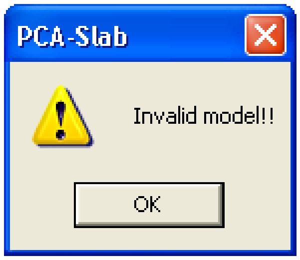


Figure 3-39 Invalid Model Error Message

The missing data must be completed before execution. A status window pops up and shows the status during the execution. If the analysis and/or design are completed successfully, a dialog box similar to Figure 3-40 is shown. If the execution is not successful, an error message will be shown and the execution is terminated.

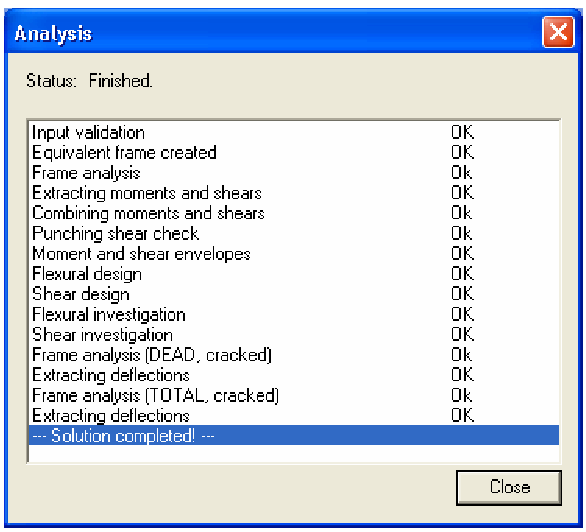


Figure 3-40 Successful Execution Dialog Box

Report

The **Report** command shows a report of analysis and/or design as shown in Figure 3-41. This command is not available before the analysis and/or design are performed.

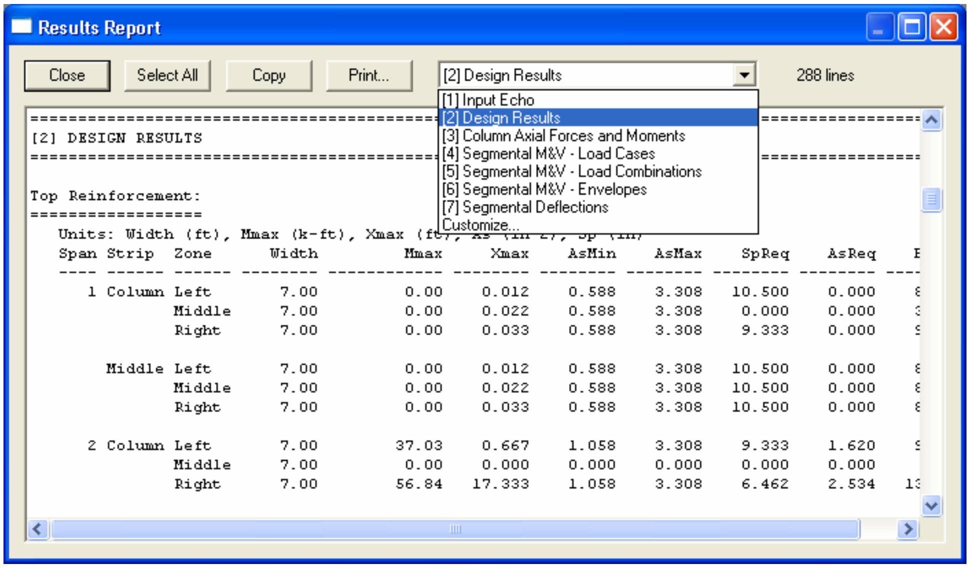


Figure 3-41 View Results Window

View Menu

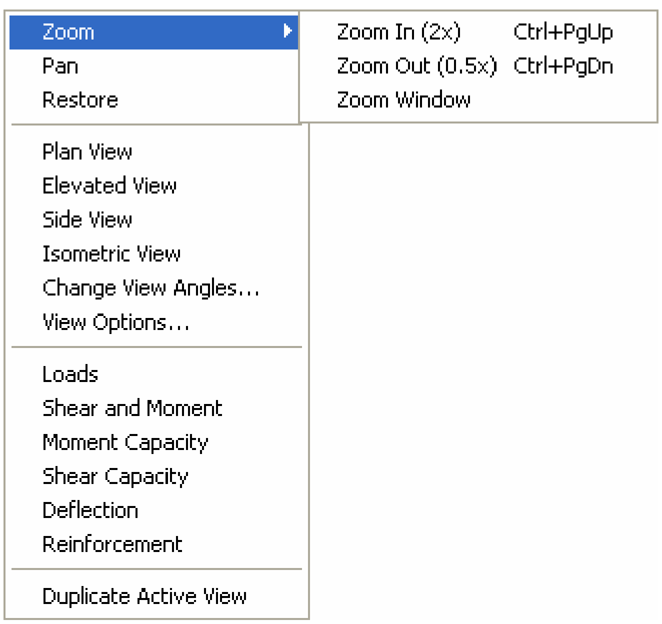


Figure 3-42 View Menu

The **View** menu commands enable you to modify the floor system's appearance on the screen to suit your viewing needs and enable you to view the result diagrams. The **View** menu contains the following commands: **Zoom**, **Pan**, **Restore**, **Plan View**, **Elevated View**, **Side View**, **Isometric View**, **Change View Angles**, **View Options**, **Loads**, **Shear**

and Moment, **Moment Capacity**, **Shear Capacity**, **Reinforcement**, **Deflection** and **Duplicate Active View**.

Zoom

The **Zoom** menu contains a cascade sub-menu, which enables you to zoom in and out on any portion of your floor system. Select Window from the sub-menu and use the mouse to specify a zooming region; the program will enlarge the portion you select. Select the **In(2x)** or **Out(0.5x)** to enlarge or reduce the model by two times, respectively.

Pan

The **Pan** command allows you to move your model on the plane of the screen. You may move the model in any direction. The mouse curser is changed to a palm shape once the **Pan** command is selected. Press and hold the left mouse button on the view window and drag to the new location. After the mouse button is released, the model is moved in the same distance and direction as the mouse curser from the original position.

Restore

The **Restore** command will redraw the floor system in full size. If you have altered your screen view using the **Zoom** command, select **Restore** to restore the figure's original proportions.

Plan View

Select **Plan View** command to show the plan view window.

Elevated View

Select **Elevated View** command to show the elevated view window.

Side View

Select **Side View** command to show the side view window.

Isometric View

Select **Isometric View** command to show the isometric view window.

Change View Angles

The **Change View Angle** command allows you to modify the angle at which the floor system is displayed in the Isometric View. The default angles are set at -45 about the X axis and 45 about the Z axis. A more convenient way to change the view angle is to use the keyboard short cut **Ctrl + Arrow Keys**. To rotate around Z axis, press **Ctrl + ←** or **Ctrl + →**. To rotate around X axis, press **Ctrl + ↑** or **Ctrl + ↓**.

View Options

The **View Options** command allows you to view selected members of the floor system in the view windows as shown in Figure 3-43. Clicking the left mouse button on the check boxes next to the items in the dialog box, or tabbing to the member type and pressing the space bar will toggle the selection. **pcaSlab** will draw any members that contain a ✓ in the box.

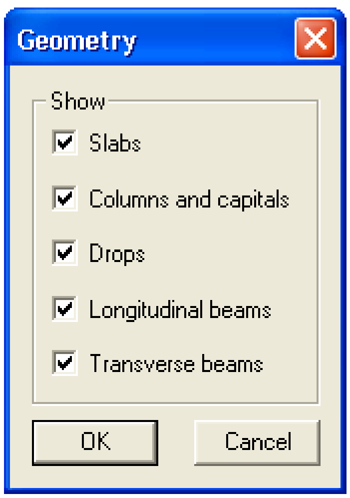


Figure 3-43 Geometry Dialog Box

Loads

Select **Loads** command to show the load view window.

Shear and Moment

Select **Shear and Moment** command to show the shear and moment diagram view window. The analysis and/or design must be performed before selecting this command. Otherwise Figure 3-44 will be shown instead.

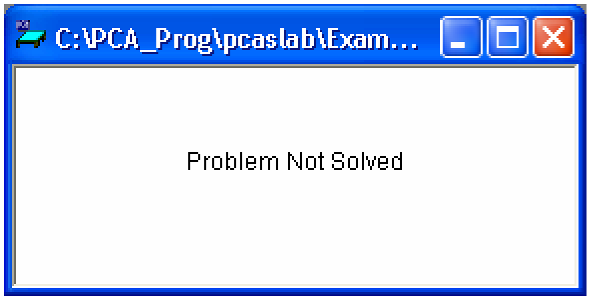


Figure 3-44 Problem Not Solved Window

Moment Capacity

Select **Moment Capacity** command to show the moment capacity diagram view window. The analysis and/or design must be performed before selecting this command. Otherwise Figure 3-44 will be shown instead.

Shear Capacity

Select **Shear Capacity** command to show the shear capacity diagram view window. The analysis and/or design must be performed before selecting this command. Otherwise Figure 3-44 will be shown instead.

Reinforcement

Select **Reinforcement** command to show the reinforcement view window. The analysis and/or design must be performed before selecting this command. Otherwise Figure 3-44 will be shown instead.

Deflection

Select **Deflection** command to show the deflection diagram view window. The analysis and/or design must be performed before selecting this command. Otherwise Figure 3-44 will be shown instead.

Duplicate Active View

Select **Duplicate Active View** to make a copy of the current avtive view window.

Options Menu

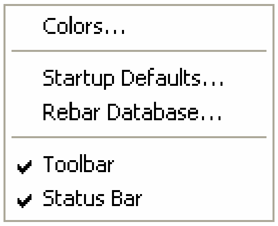


Figure 3-45 Option menu

The **Options** menu allows you to change the startup options of the **pcaSlab** program to suit your needs. The **Options** menu contains the following commands: **Colors**, **Startup Defaults**, **Rebar Database**, **Toolbar**, and **Status Bar**.

Colors

The **Colors** command allows you to change the background color, member color, load color, text color, diagram color, etc., as shown in Figure 3-46. You may save the new colors as default setting, which will be used when **pcaSlab** is executed in the future.

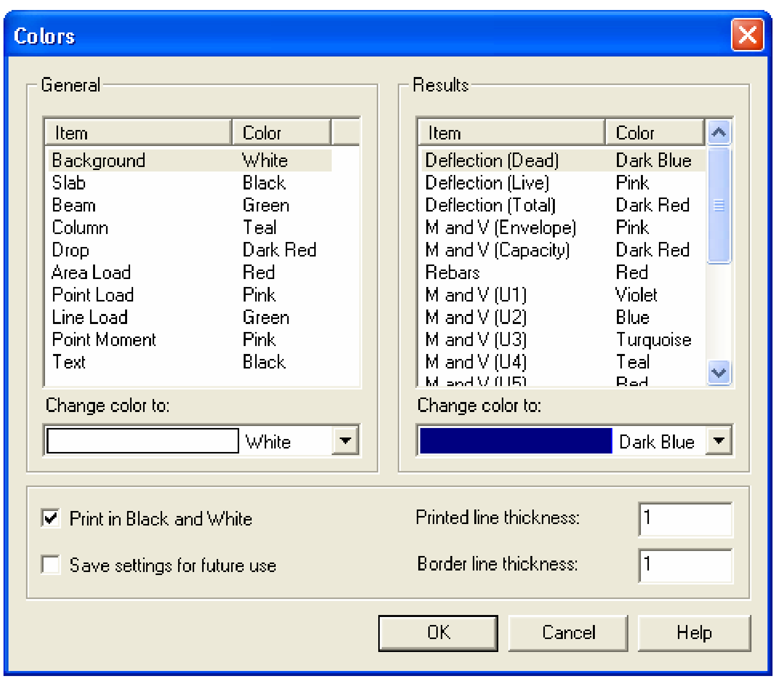


Figure 3-46 Changing Color Dialog Box

Startup Defaults

The **Startup Defaults** command allows you to enter engineer name, change the default design code, rebar database, and the data directory, a directory where **pcaSlab** looks for data when it is executed.

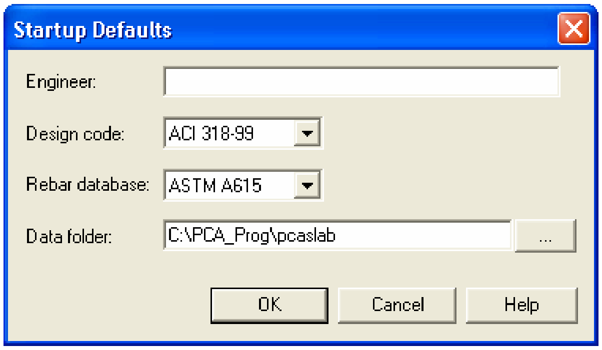


Figure 3-47 Startup Default Dialog Box

Rebar Database

The **Rebar Database** command allows you to view the pre-defined rebar information and define your own database as shown in Figure 3-48. The user-defined database can be selected from the General Information dialog box.

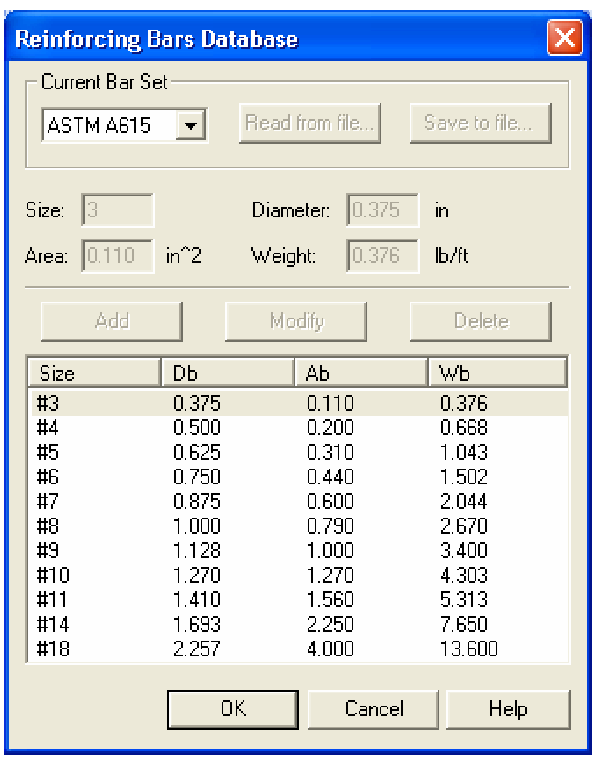


Figure 3-48 Rebar Database Dialog Box

Toolbar

Check the Toolbar command with a ✓ sign to show the tool bar. Select the command again to clear the ✓ sign to hide the tool bar. The tool bar is shown by default.

Status Bar

Check the Status Bar command with a ✓ sign to show the status bar. Select the command again to clear the ✓ sign to hide the status bar. The status bar is shown by default.

Window Menu

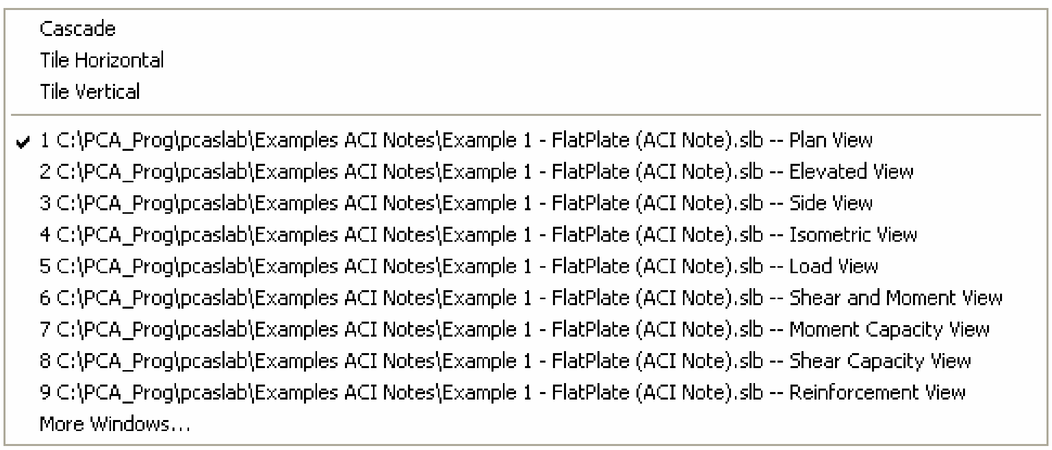


Figure 3-49 Window Menu

This menu enables you to arrange view windows shown on screen. The **Window** menu contains the following commands: **Cascade**, **Tile Horizontal** and **Tile Vertical**.

Cascade

The **Cascade** command displays all the open windows in the same size, arranging them on top of each other so that the title bar of each is visible. The current active view widow will be on the top after the execution of the **Cascade** command.

Tile Horizontal

The **Tile Horizontal** command arranges all open windows horizontally so that no window overlaps another. The current active view widow will be on the most left or on the upper-left corner of the screen after the execution of the **Tile Horizontal** command.

Tile Vertical

The **Tile Vertical** command arranges all open windows vertically so that no window overlaps another. The current active view widow will be on the most left or on the upper-left corner of the screen after the execution of the **Tile Vertical** command.

Remaining Commands

The remaining menu items are in a list of the windows that are available for viewing. Selecting any window from this menu will restore the window to its previous size and position from an icon.

Help Menu

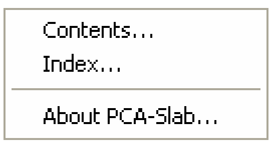


Figure 3-50 Help menu

The **Help** menu includes commands that enable you to obtain online help for the **pcaSlab** program and show the copyrights and registration information about your software. The Help menu contains the following commands: **Index** and **About pcaSlab**.

Contents

The **Contents** command shows the electrical version of this manual in a tri-pane window. The top pane contains the commonly used control buttons. The left pane contains the index of all the help topics. The contents of a topic can be shown in the right pane by double clicking the topic on the left pane. More information on the help system can be obtained from the Microsoft Windows manual.

Index

The **Index** command shows the electrical version of this manual in a tri-pane window. The top pane contains the commonly used control buttons. The left pane contains the table of contents of all the help topics. The contents of a topic can be shown in the right pane by single clicking the topic on the left pane. More information on the help system can be obtained from the Microsoft Windows manual.

About pcaSlab

The **About pcaSlab** command displays a dialog box that gives you copyright information, the version number of the **pcaSlab** program, your firm name, city, and state.

The Control Menu

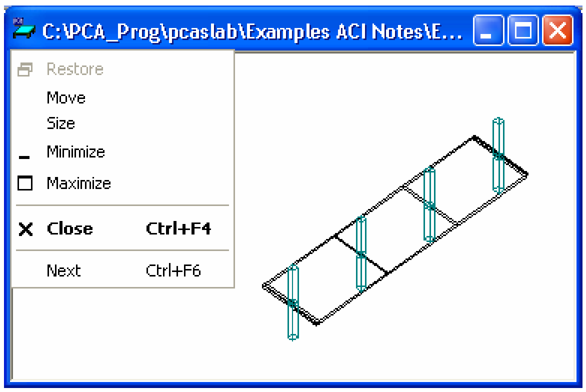


Figure 3-51 Control Menu

All the view windows have one pull-down menu located in the upper left hand corner of the open window. This is the **Control** menu. To access the **Control** menu, press **Enter + F6** to cycle through the windows and press **Alt + –** (hyphen), to open the Control menu of the desired window. The following is a list and a brief description of the commands in this menu.

Restore

The **Restore** command will restore a window or an icon to its previous size and position. This menu item is available when the window is iconized or maximized.

Move

The **Move** command moves the window to a new location. Select **Move** and use the arrow keys to move the window in the desired direction and select **Enter** to accept the new location.

Size

The **Size** command resizes a window. Select **Size** and use the arrow keys to move the border of the window in the desired direction and select **Enter** to accept the new size.

Minimize

The **Minimize** command reduces a window to an icon and positions it at the bottom of the screen.

Maximize

The **Maximize** command enlarges a window to fit your entire screen.

Close

The **Close** command is used to close a window and return it to an icon and the bottom of the screen.

Next

The **Next** command switches among open windows and icons.

Program Toolbar

Toolbar

Figure 3-52 Program Toolbar

 Close the current data file if there is one and start a new data file. The equivalent menu command is **File/New**.

 Open an existing data file on hard disk. The equivalent menu command is **File/Open**.

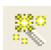
 Save the current data file to hard disk. The equivalent menu command is **File/Save**. If you have not changed the default file name (**pcaSlab**1, **pcaSlab**2, etc.), the equivalent menu command is **File/Save As**.

 Print view window. The equivalent menu command is **File/Print View**.

 Print results. The equivalent menu command is **File/Print Results**.

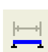
Copy Bitmap to clipboard. The bitmap then can be pasted to a word processing or presentation software such as Microsoft Word or Microsoft PowerPoint.

Copy Metafile to clipboard. The metafile then can be pasted to a word processing or presentation software such as Microsoft Word or Microsoft PowerPoint.

 Open the Data Input Wizard. The Data Input Wizard will guide you to enter the necessary input to your project. The equivalent menu command is **Input/Data Input Wizard**.

 Enter general information. The equivalent menu command is **Input/General Information**.

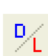
 Enter material properties. The equivalent menu command is **Input/Material Properties**.

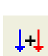
 Enter span geometry information for slabs, longitudinal beams, and ribs. The equivalent menu command is **Input/Spans**.

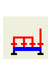
 Enter support information for columns, drop panels, column capitals, and transverse beams. The equivalent menu command is **Input/Support**.

 Enter reinforcement criteria for slab and ribs, and beams. The equivalent menu command is **Input/Reinforcement Criteria**.

 Enter reinforcing bar information for column strips, middle strips, beams, and beam stirrups. This button is disabled if **Design** run mode is selected from the **General Information** dialog box. The equivalent menu command is **Input/Reinforcing Bars**.

 Enter load cases. The equivalent menu command is **Input/Load Cases**.

 Enter load combinations. The equivalent menu command is **Input/Load Combinations**.

 Enter span loads. The equivalent menu command is **Input/Span Loads**.

 Enter lateral effects. The equivalent menu command is **Input/Lateral Effects**.

 View plan geometry. The equivalent menu command is **View/Plan View**.

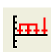
 View elevated geometry. The equivalent menu command is **View/Elevated View**.

 View side geometry. The equivalent menu command is **View/Side View**.

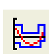
 View isometric geometry. The equivalent menu command is **View/Isometric View**.

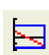
 Execute the analysis and/or design. The equivalent menu command is **Solve/Execute**.

 View results. The equivalent menu command is **Solve/Report**.

 View loads. The equivalent menu command is **View/Loads**.

 View shear and moment of the whole geometry or a single span. The equivalent menu command is **View/Shear and Moment**.

 View moment capacity of the whole geometry or a single span. The equivalent menu command is **View/Moment Capacity**.

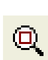
 View shear capacity of the whole geometry or a single span. The equivalent menu command is **View/Shear Capacity**.

 View deflection of the whole geometry or a single span. The equivalent menu command is **View/Deflection**.

 View flexure reinforcement for beam strip, middle strips, and column strips. View shear reinforcement for beam strips. The equivalent menu command is **View/Reinforcement**.

 Zoom in view window to magnify the system. The equivalent menu command is **View/Zoom/In(2x)**.

 Zoom out view window to reduce the system. The equivalent menu command is **View/Zoom/Out(0.5x)**.

 Zoom any part of a view window. The equivalent menu command is **View/Zoom/Window**.

 Move the model in the screen plane. The equivalent menu command is **View/Pan**.

 Restore a view window. The equivalent menu command is **View/Redraw**.

Program Input Wizard

Input Wizard

The Data Input Wizard is designed to make the inputting process easier. By selecting the **Data Input Wizard** from **Input** Menu or selecting  from the tool bar, a logical sequence of dialog boxes will automatically be displayed allowing you to enter data for your floor system.

Program Input Dialog Windows

Input Dialog Windows

A total of ten steps may be needed to enter data for a model if all of them are applicable. They are General Information, Material Properties, Spans, Supports, Reinforcing Criteria, Load Cases, Load Combinations, Span Loads, and Lateral Effects. If **Data Input Wizard** is selected from the **Input** menu, you will be guided through these input dialog boxes one by one, automatically.

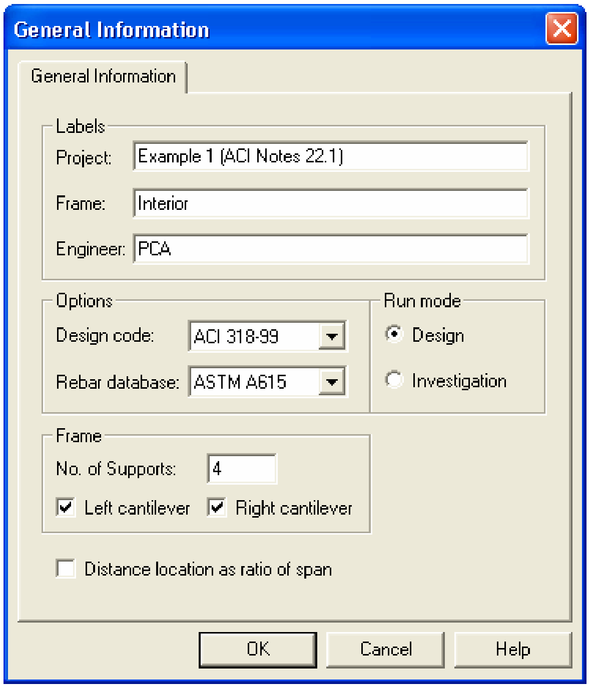


Figure 3-53 General Information Dialog Box

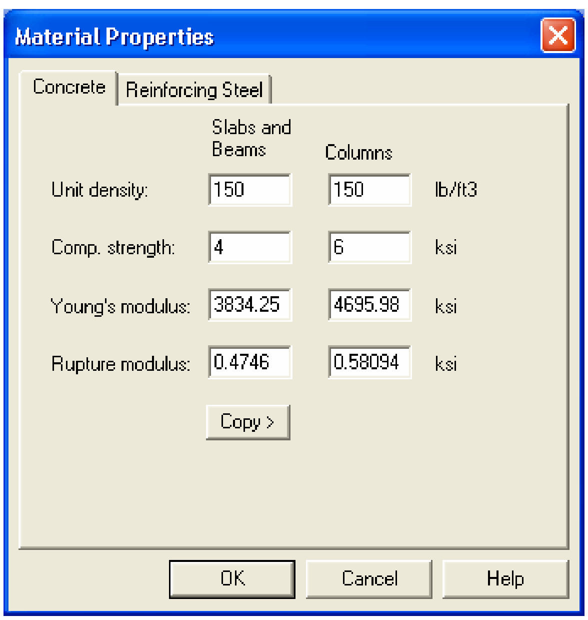


Figure 3-54 Material Properties Dialog Box

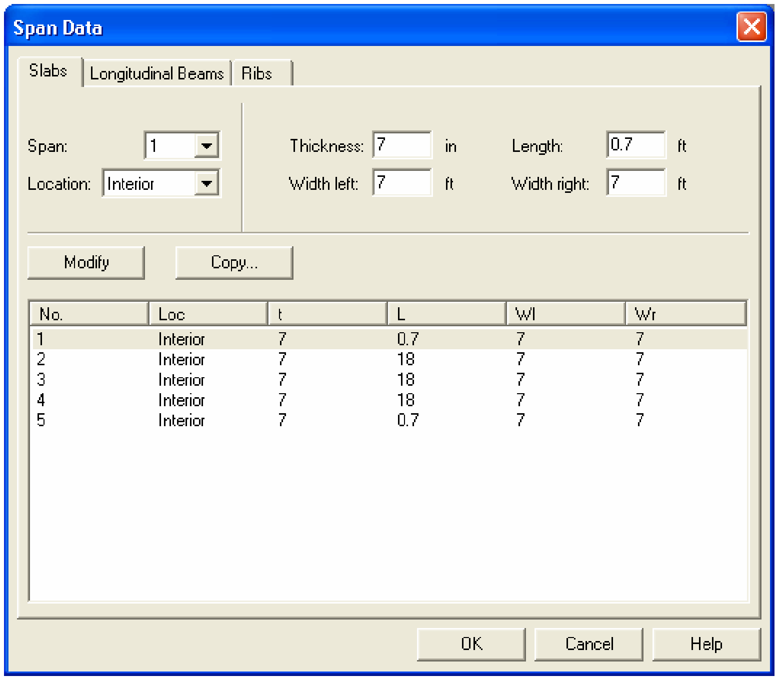


Figure 3-55 Spans Dialog Box

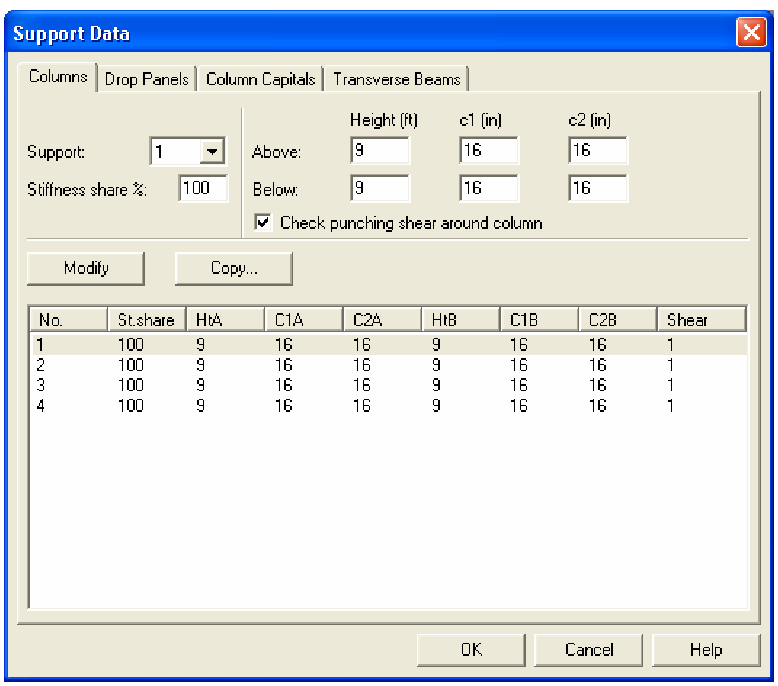


Figure 3-56 Supports Dialog Box

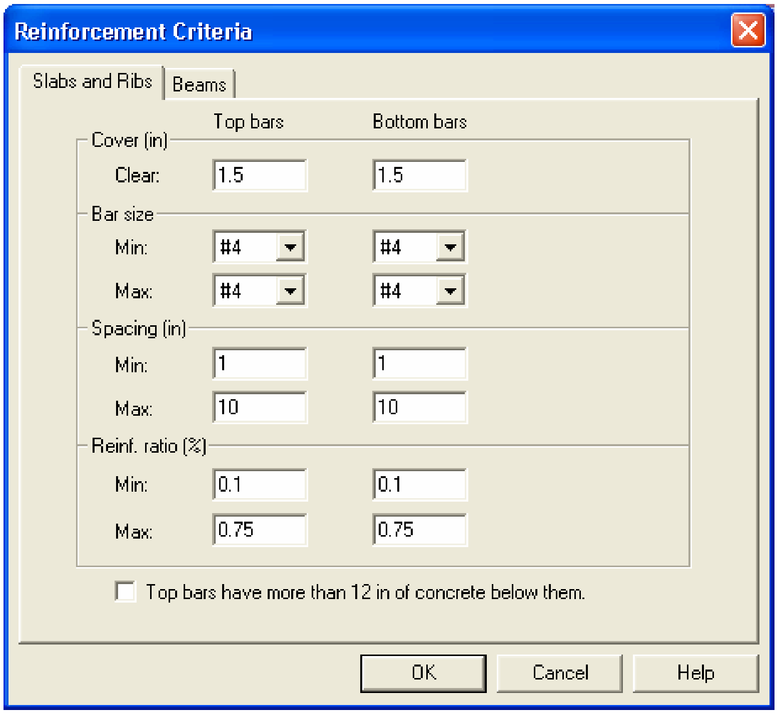


Figure 3-57 Reinforcing Criteria Dialog Box

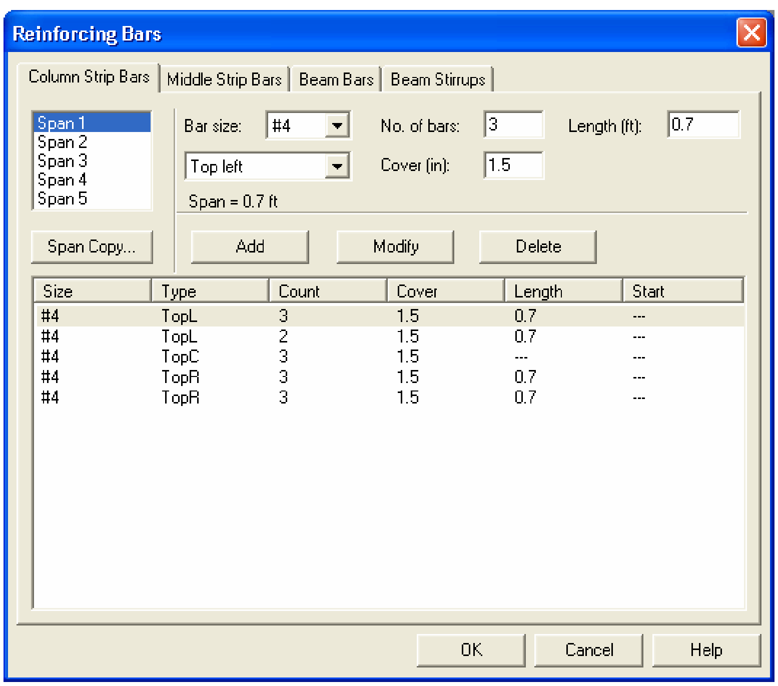


Figure 3-58 Reinforcing Bars Dialog Box

Figure 3-59 Load Cases Dialog Box

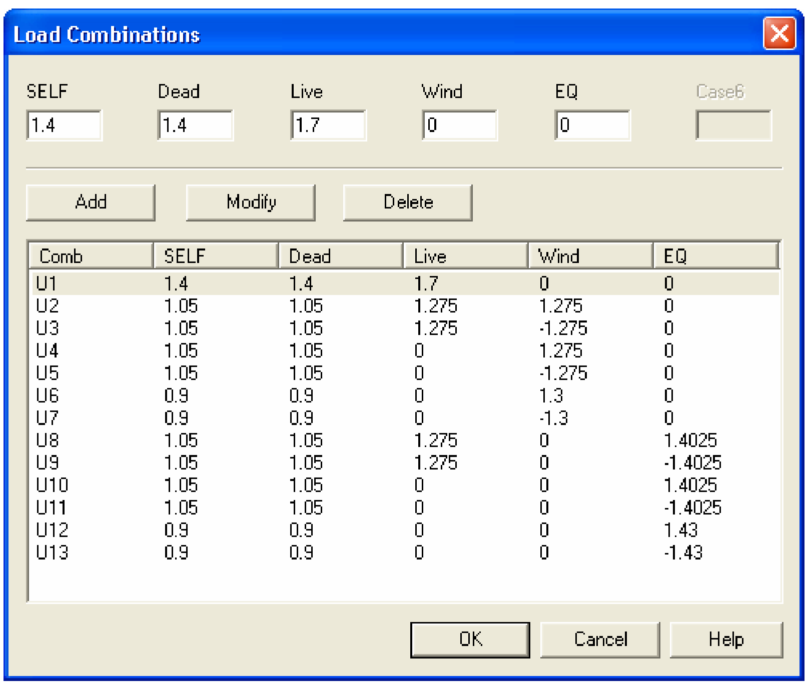


Figure 3-60 Load Combinations Dialog Box

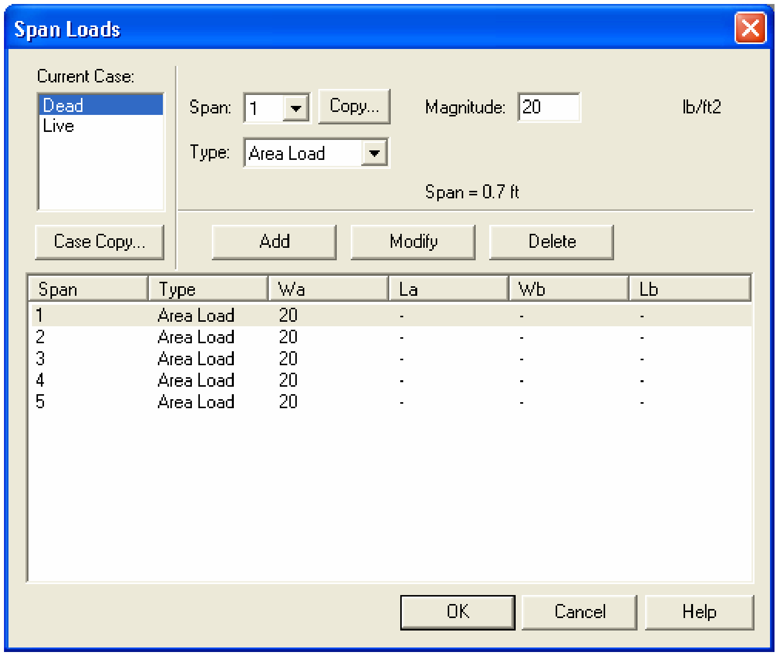


Figure 3-61 Span Loads Dialog Box

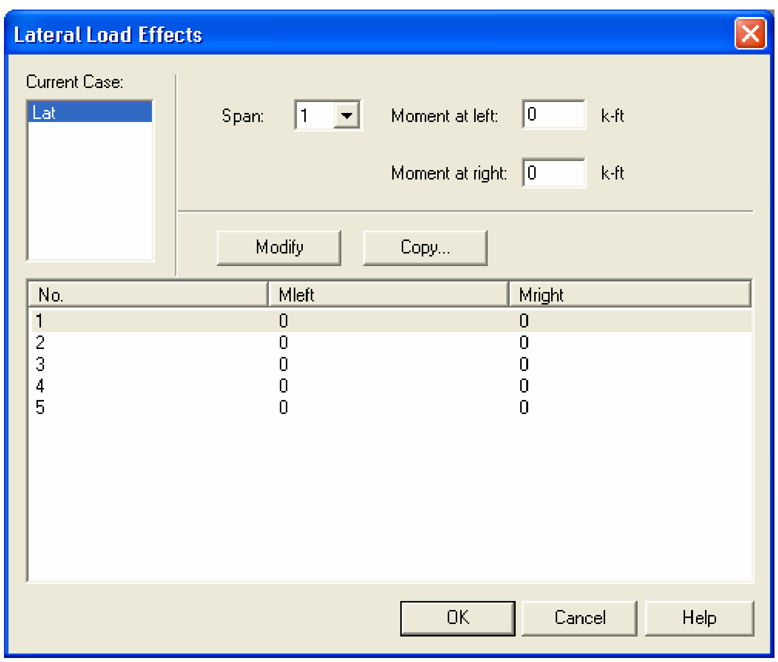


Figure 3-62 Lateral Effects Dialog Box

Program Output Dialog Windows

Output Dialog Windows

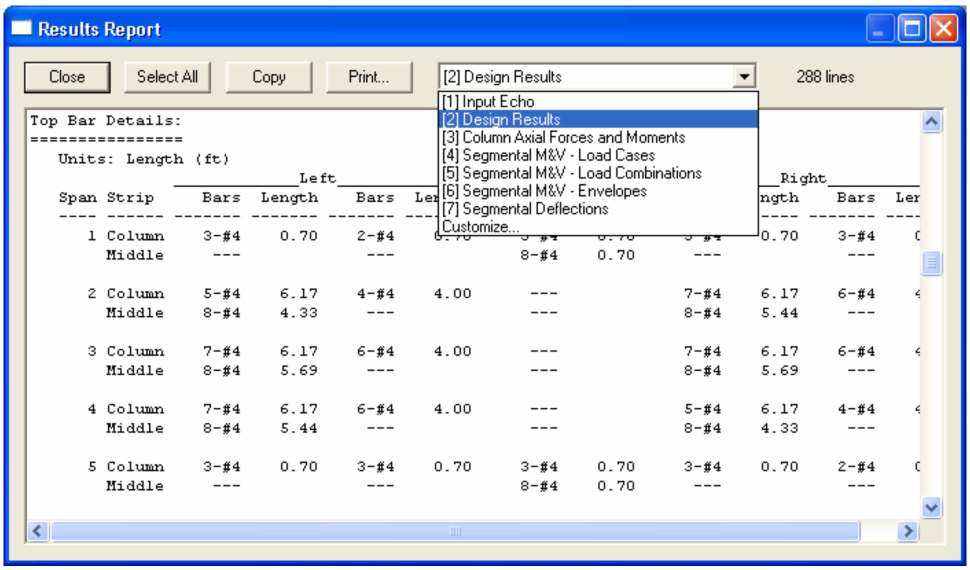


Figure 3-63 Result Report

Ch. 4 Operating the Program

Working with Data Files (menu File)

Creating a New Data File

When you first load **pcaSlab** you will have a new file ready for input. The data will not have a filename associated with it, therefore, *"pcaSlab1"* will appear in the title bar. If a new data file is created after the *"pcaSlab1"*, the new data file will be named as *"pcaSlab2"*, and so on.

To start a new data file:

1. If you are already in the program and in an existing file, select **New** button  or **New** menu command to clear your screen and return you to the default settings. If existing data has been changed prior to executing the **New** command, **pcaSlab** will ask if you would like to save the data.
2. After **New** is selected**, Auto Input** command in the Input menu may be used. This command guides you through the inputting process by automatically displaying all the dialog boxes necessary to design your floor system. You may cancel the auto input mode by selecting **Cancel** button from any dialog box.
3. After you enter data through the Input menu, use the **Save As** command to give the file a name.

Opening Existing Data File

pcaSlab allows you to open data files that were saved at an earlier time.

To open an existing data file:

1. Select **Open** command from the File menu or click the **Open** button  to bring in an existing **pcaSlab** data file. The dialog box of Figure 4-1 will be displayed. All the files with an .slb extension contained in the current drive and directory will be displayed in the list box. This dialog box also enables you to change the current drive and directory.

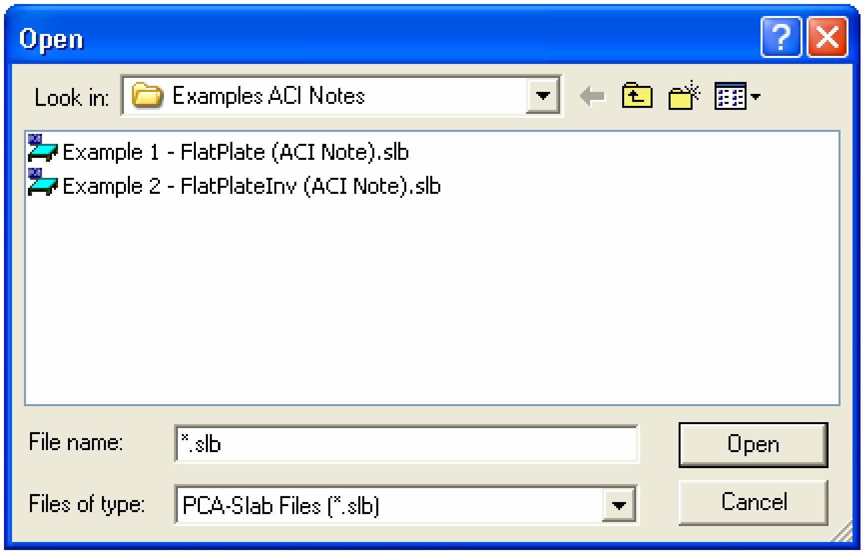


Figure 4-1 Open Dialog Box

2. Type in the name of the file you want to open. You may also select a file from the provided list with the mouse or using the keyboard by tabbing to the list and using the up and down arrow keys.
3. Press **Open** to exit the dialog box and allow **pcaSlab** to read the data. You may combine steps 2 and 3 of this procedure by double clicking the left mouse button over the desired file from the provided list.

Importing ADOSS Data File

pcaSlab allows you to open ADOSS data files that were saved at an earlier time.

To open an existing ADOSS data file:

1. Select **Open** command from the File menu or click the **Open** button  to bring in an existing ADOSS data file. The dialog box of Figure 4-2 will be displayed. All the files with an .ads extension contained in the current drive and directory will be displayed in the list box. This dialog box also enables you to change the current drive and directory.

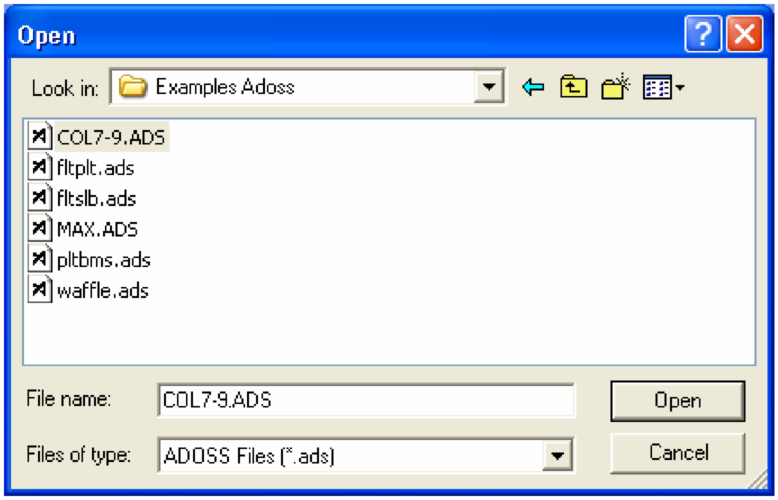


Figure 4-2 Importing ADOSS data file Dialog Box

2. Type in the name of the file you want to open. You may also select a file from the provided list with the mouse or using the keyboard by tabbing to the list and using the up and down arrow keys.
3. Press **Open** to exit the dialog box and allow **pcaSlab** to read the data. You may combine steps 2 and 3 of this procedure by double clicking the left mouse button over the desired file from the provided list.

Note:

1. The extension name of an ADOSS file is .ADS, while the extension name of a **pcaSlab** file is .SLB.
2. Only two-way system models developed in ADOSS are supported by **pcaSlab**.

Saving the Data File

To save your data with the same filename:

1. Select the **Save** command from the File menu or simply click the **Save** button . If the data has not been modified since the last save or **Save As** command was executed, this option will not be available. The file will be updated and the **Save** command and button will be shaded gray.

To give your data a new filename:

1. If you have not saved your data yet, select the **Save** or **Save As** command from the File menu. Either command will have the same effect. If you would like to save a data file that currently has a filename with a new filename, select the **Save As** command. The dialog box of Figure 4-3 will appear. When no filename has been given to the current data, the default filename is "**pcaSlab**1.slb", and it

is highlighted in the edit box.

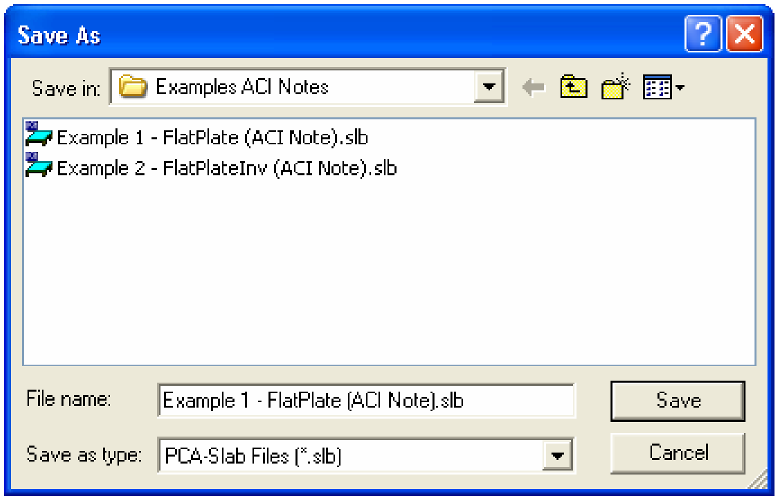


Figure 4-3 Save As Dialog Box

2. Type a new name to overwrite the current name.
3. Press **Save** to exit the dialog box and save the data into the filename specified.

Most Recently Used Files (MRU)

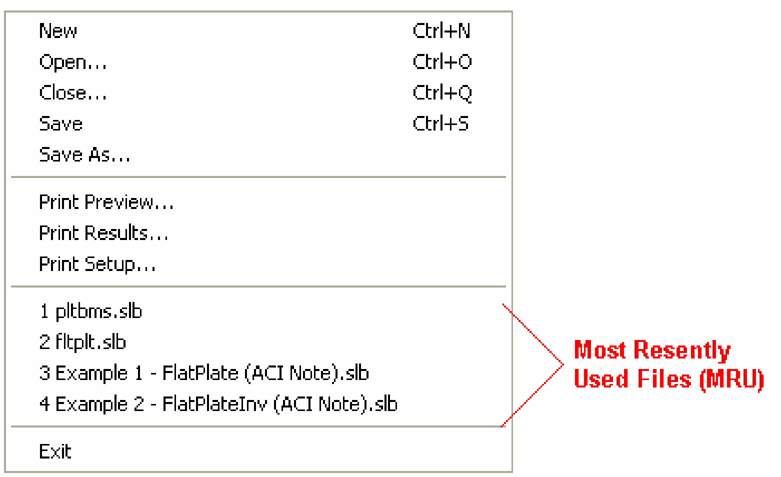


Figure 4-4 Most Recently Used File List (MRU)

The Most Recently Used Files (MRU) list shows the four data files that were opened most recently. Selecting a data file from this list makes it easier and faster to open the file. The list is empty when the program is executed for the first time.

Specifying the Model Data (menu Input)

Data Input Wizard

Data Input Wizard

The Data Input Wizard is designed to make the inputting process easier. By selecting the **Data Input Wizard** from **Input** Menu or selecting  from the tool bar, a logical sequence of dialog boxes will automatically be displayed allowing you to enter data for your floor system.

General Information

Defining General Information

The **General Information** command allows you to enter labels, design code, rebar database, run mode, and the frame information needed by **pcaSlab** to proceed with the input process. You must choose this command before doing any further inputting since this command affects the availability of the commands in the Input menu.

To enter general information:

1. Select the **General Information** command from the Input menu or click the  button from the tool bar. The dialog box of Figure 4-5 will appear.

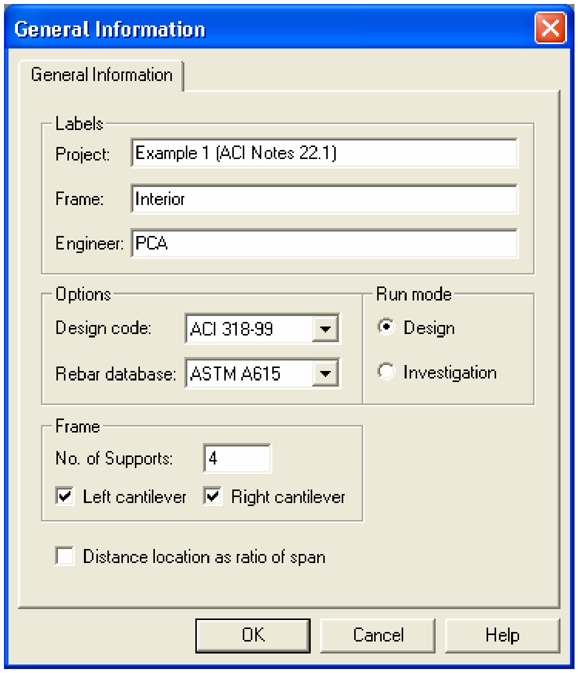


Figure 4-5 General Information Dialog Box

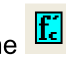
2. Enter the project name, frame name, and engineer name in the **Label** frame box.
3. Select the building standard you want your floor system to be designed to (ACI 318-99, ACI 318M-99, CSA-A23.3-94, CSA-A23.3-94E) from the **Option** frame box**.**
4. Select the **Design** or **Investigation** from the **Run mode** frame box.
5. In the **Frame** frame box, enter the Number of supports of the frame. The default number of supports is 2. The minimum and maximum number of spans are 1 and 20 spans, respectively. Therefore the minimum number of supports is 2 and the maximum number of supports is 21.
6. Check the **Left cantilever** and/or **Right cantilever** check boxes if left cantilever and/or right cantilever exists in the frame respectively.
7. Check the **Distance location as ratio of span** if the locations of loads need to be entered as a ratio of the length of a span.
8. Press **Ok** button to exit the dialog box and allow **pcaSlab** to use the new data. If using the Auto Input, click the Next button to the next dialog box.

Material Properties

Defining Material Properties

The **Material Properties** command from the Input menu allows you to input material properties of the concrete and the reinforcement. There are two tabs in this dialog box. One is for concrete and the other is for reinforcing steel. This command must be executed in order to perform a design of the floor system. Use the tab key to get to each edit box then type in your values, or use your mouse and click directly on the desired tab and box, then type in your values. Refer to "Material Properties" for a detailed explanation of the default values.

To define material properties:

1. Select the **Material Properties** command from the Input menu or click the  button on the tool bar. The dialog box of Figure 4-6 will appear.

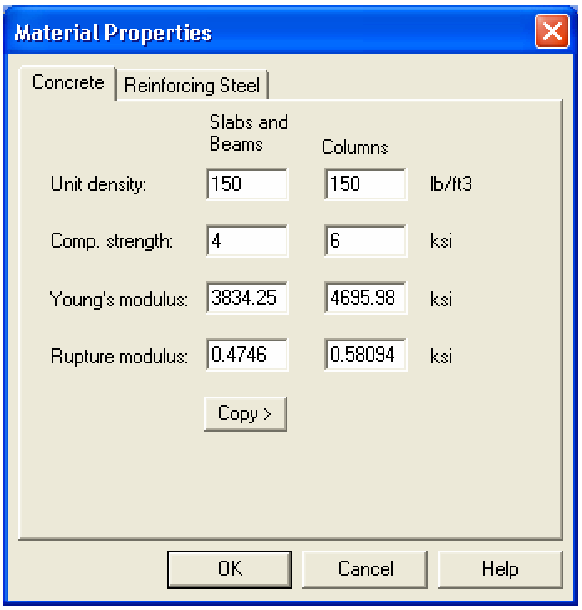


Figure 4-6 Concrete Properties Dialog Box

2. Click the **Concrete** tab and enter the concrete density for the following members: Slabs, Beams, and Columns.
3. Enter the concrete compressive strength. By entering a value for the compressive strength, values for Young's modulus and rupture modulus will automatically be computed for the slabs, beams, and columns. Young's modulus and rupture modulus will automatically be shown in the corresponding text boxes.
4. If you have values for the rupture modulus, enter the values in the text boxes for the slabs, beams, and columns. Default values are computed based on average split tensile strength of concrete, which is calculated internally by the program. These values will be used for deflection analysis. A large value for the rupture modulus will produce a deflection analysis based on gross, non-cracked, sections. The CSA A23.3-94 Standard requires that for the calculation of slab deflections a rupture modulus value equal to $0.6\sqrt{f_c'}\,/\,2$ be used. **pcaSlab** defaults to this value of the rupture modulus for the slab concrete in CSA design runs.
5. Click the Reinforcing **Steel** tab as shown in Figure 4-7.

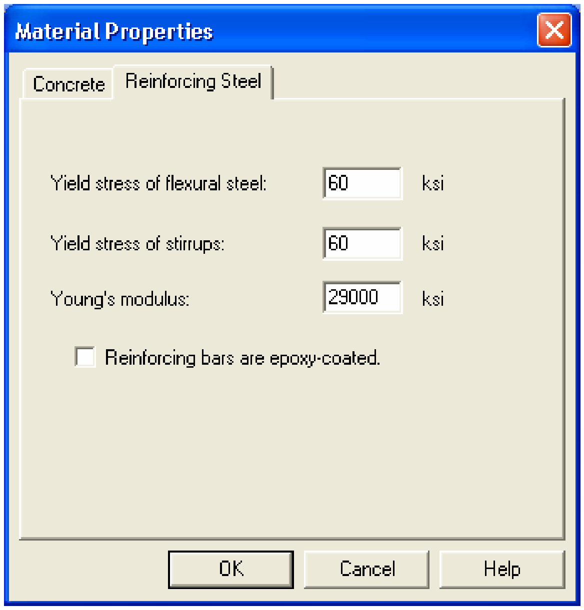


Figure 4-7 Reinforcing Steel Properties dialog box

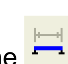
6. Enter the yield stress of flexure steel.
7. Enter the yield stress of stirrups.
8. Enter the Young's modulus for flexural steel and stirrups.
9. Select whether the main reinforcement is epoxy-coated by clicking the left mouse button on the box or tabbing to the box and pressing the space bar. This selection affects development lengths.
10. Press **Ok** button to exit the dialog box so that **pcaSlab** will use these material properties. If using the Auto Input, click the **Next** button to the next dialog box.

Spans

Defining the Slabs

The **Spans** command from the Input menu is available for all floor systems. Span numbers, which are determined from the number of supports entered in the General Information box, are automatically filled into the **Span** drop-down list in the Span Data dialog box.

To input slab geometry:

1. Select the **Spans** command from the Input menu or click the  button on the tool bar. Click the left mouse button on the Slabs tab. The dialog box of Figure 4-8 will appear.

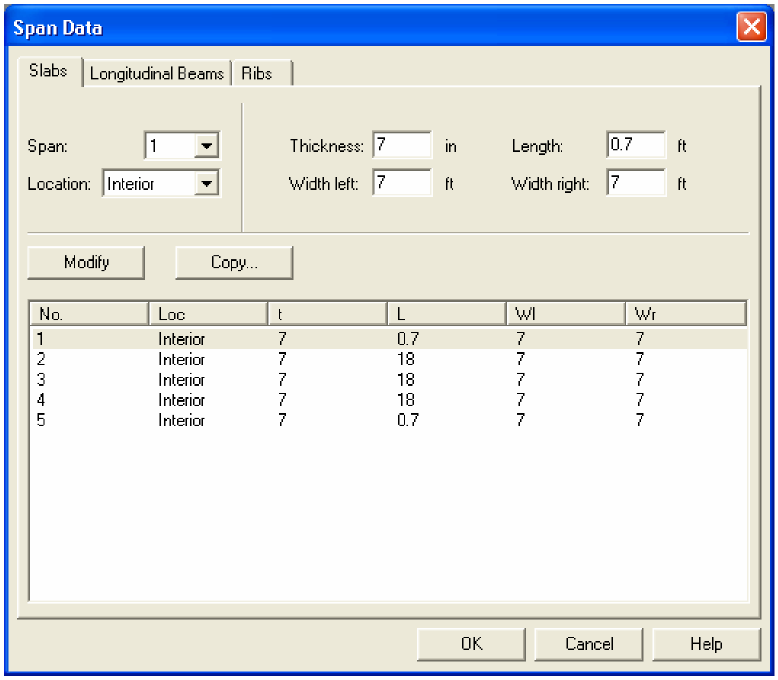


Figure 4-8 Defining the Slabs dialog box

2. Select the number of the span, for which dimensions will be entered, from the **Span** drop-down list.
3. Select the span location from the **Location** drop-down list. Three types of locations are available: Interior, Exterior Left, and Exterior Right. The "left" and "right" are defined as you look along the direction of analysis. If a span has design strips on both sides it should be an "Interior" span. If a span has only a left design strip, it should be an "Exterior Left" span. If a span has only a right design strip, it should be an "Exterior Right" span.
4. Enter the slab thickness of the span.
5. Enter the span length from column centerline to column centerline or edge to column centerline for the two cantilever spans in the **Length** edit box. If the program detects a cantilever span length less than one-half the column dimension in the direction of analysis, an error message will pop up when the frame is analyzed. If a partial load is affected by the span length, a message warns the user of this condition.
6. Enter the span design width in the transverse direction of analysis on the left and right side of the column (see Figure 4-9). These distances are usually one-half the distance to the next transverse column or edge of the slab for exterior spans. The left and right designations are arbitrary. Both interior and exterior spans may be used in a design strip. An exterior width will automatically be designated by **pcaSlab** by entering a width value less than or equal to the transverse column dimension. Exterior sides do not contribute to the attached torsional stiffness, although they do contribute to loading. **pcaSlab** will use the total width entered for weight and superimposed loading but will use code allowed dimensions for flange width and stiffness computations.

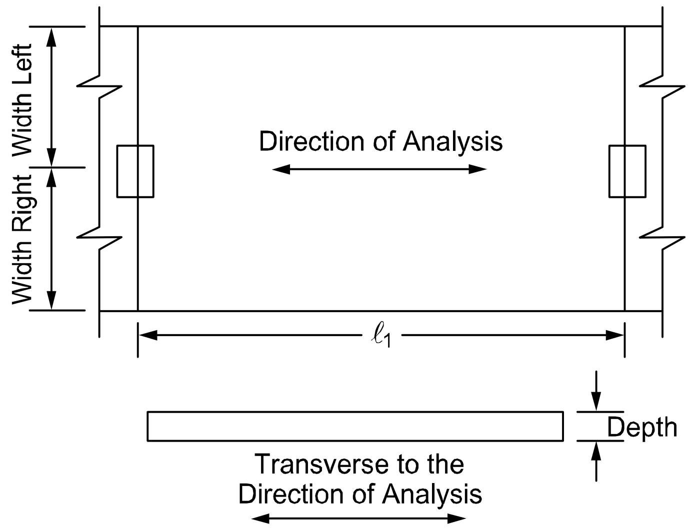


Figure 4-9 Required Slab Dimensions

7. Press the **Modify** button to update the slab geometry.
8. Repeat steps 2 through 7 until all the spans have been updated. You can use the **Copy** button as a shortcut.
9. Press **Ok** button to exit the dialog box and allow **pcaSlab** to use the updated slab geometry.

Defining the Longitudinal Beams

Longitudinal beam dimensions are required for the beam-supported slab. Span numbers, which are determined from the number of supports entered in the General Information box, are automatically filled into the Span drop-down list.

To input geometry for beam-supported slab system:

1. Select the **Spans** command from the Input menu or click the button on the tool bar. Click the left mouse button on the **Longitudinal Beams** tab. The dialog box of Figure 4-10 will appear.

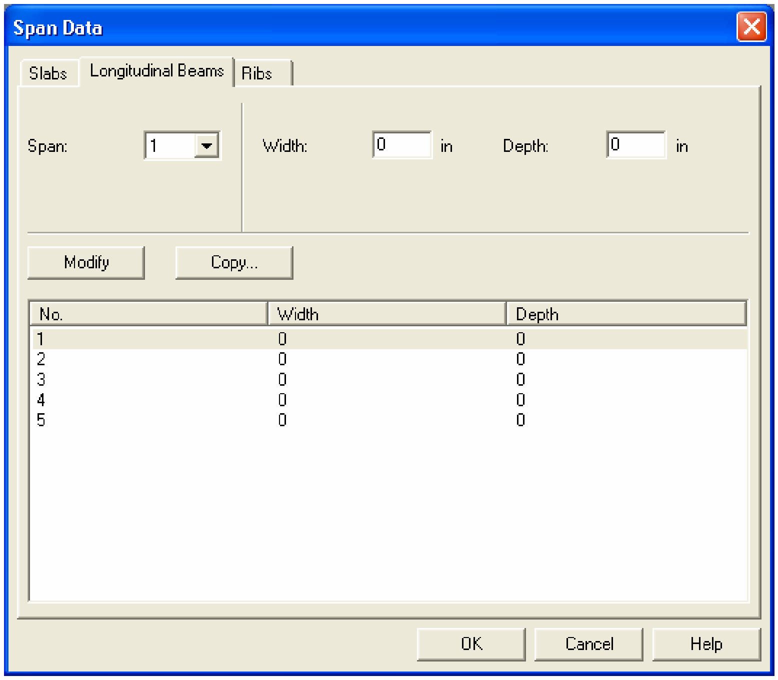


Figure 4-10 Longitudinal Beam Geometry Dialog Box

2. Select the span number from the **Span** drop-down list.
3. Enter the width of the beam (Figure 4-11).

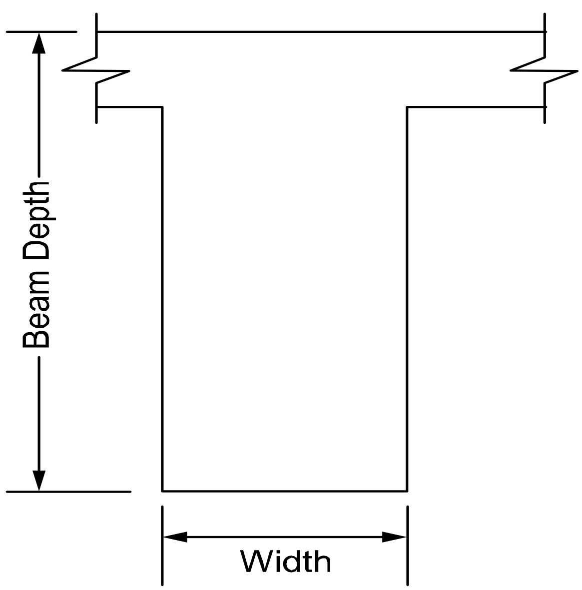


Figure 4-11 Required Longitudinal Beam Dimensions

4. Enter the depth of the beam from the top of the slab (Figure 4-11).
5. Press the **Modify** button to update the longitudinal beam geometry.
6. Repeat steps 2 through 6 until all the beams have been updated. You can use

the **Copy** button as a shortcut.

7. Press **Ok** button to exit the dialog box so that **pcaSlab** will use the new beam geometry.

Defining the Ribs

For the waffle slab system, you must define the rib geometry. The ribs are assumed to be the same throughout the strip.

To enter rib geometry:

1. Select the **Spans** command from the Input menu or click the  button on the tool bar. Click the left mouse button on the **Ribs** tab. The dialog box of Figure 4-12 will appear.

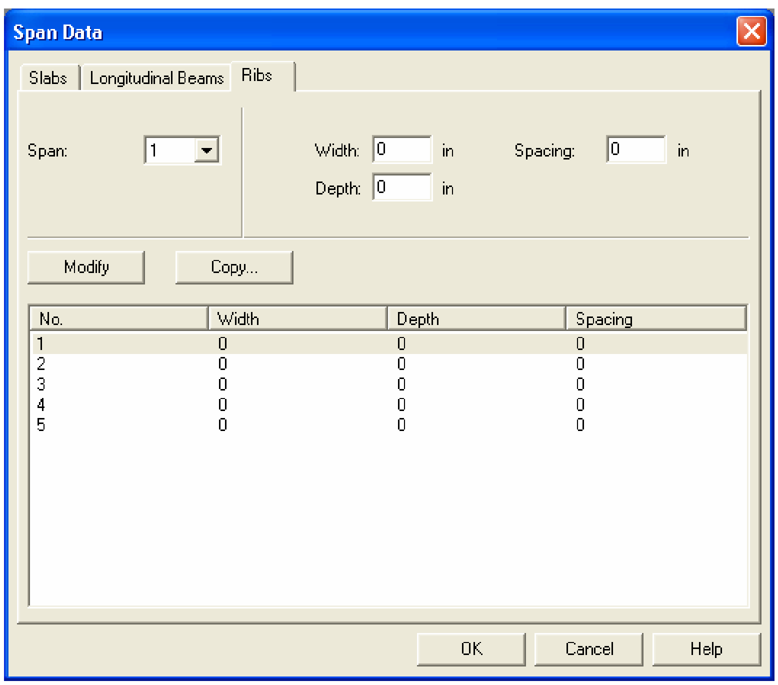


Figure 4-12 Ribs Geometry Dialog Box

2. Select the span number from the **Span** drop-down list.
3. Enter the spacing between ribs at the bottom for clear rib spacing (see Figure 4-13).

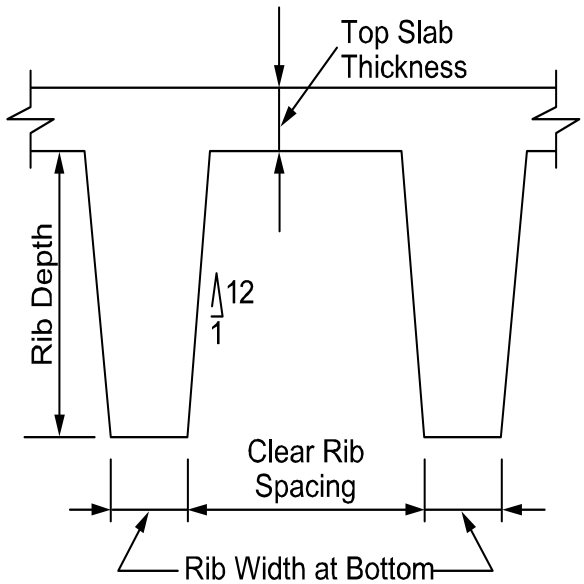


Figure 4-13 Required Rib Dimensions

4. Enter the width at the bottom for rib width (see Figure 4-13).
5. Enter the depth of the rib below the slab for Rib depth (see Figure 4-13).
6. Press **Ok** button to exit the dialog box so that **pcaSlab** can use the rib geometry.

Supports

Defining the Columns

Column data is optional. If no column is specified at the joints the joint is assumed hinged. You will be allowed to enter column dimensions above and below.

To input column/capital geometry:

1. Select the **Supports** command from the Input menu or click the  button on the tool bar. The dialog box of Figure 4-14 will appear. Click on the **Columns** tab.

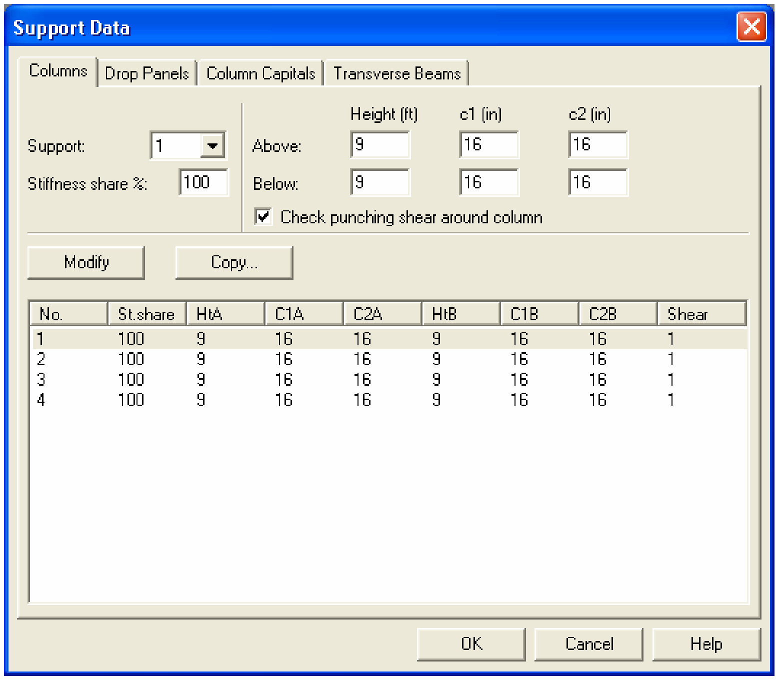


Figure 4-14 Column Geometry dialog box

2. Enter the column height above, which is the distance from the top of the design floor to the top of the floor above (see Figure 4-15). **pcaSlab** obtains the clear column height above by subtracting the average slab depth from the height given. Only the slab is considered for the floor system above. A zero dimension for the column heights above and below will create a pin condition.

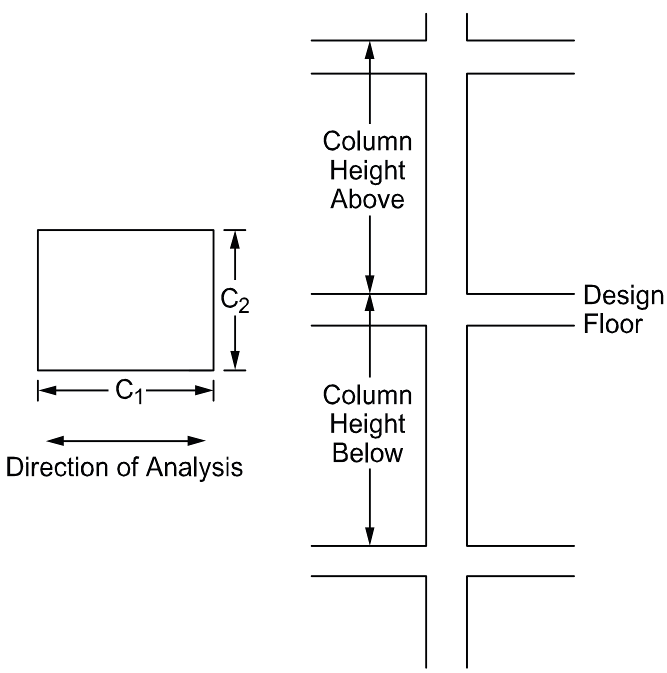


Figure 4-15 Required Column Dimensions

3. Enter the column height below, which is the distance from the design floor to the top of the floor below (see Figure 4-15). To obtain a clear column height below, the slab/drop/beam depth is subtracted from the height given. A zero dimension for the column heights above and below will create a pin condition.
4. Enter a value for c1, the column dimension in the direction of analysis (see Figure 4-15).
5. Enter a value for c2, the column dimension perpendicular to the direction of analysis (see Figure 4-15). Round columns are specified with a zero input for c2; c1 is then taken as the diameter. Zero value for Stiffness Share will create a pin condition.
6. Press the **Modify** button to update the slab geometry.
7. Check the **Check punching shear around column** if the punching shear needs to be checked.
8. Repeat steps 2 through 7 until all the columns and capitals have been updated. You can use the **Copy** button as a shortcut.
9. Press **Ok** button to exit the dialog box so that ADOSS will use the new data.

Defining the Drop Panels

Drops are only available for the flat slab or waffle slab systems and can be defined at all the support locations. The drop length and width dimensions are computed by **pcaSlab**, based on slab span dimensions, when the "Standard" is selected in the Type drop-down list.

To input drop geometry

1. Select the **Supports** command from the Input menu or click the  button then click on the **Drop** Panels tab. The dialog box of Figure 4-16 will appear.

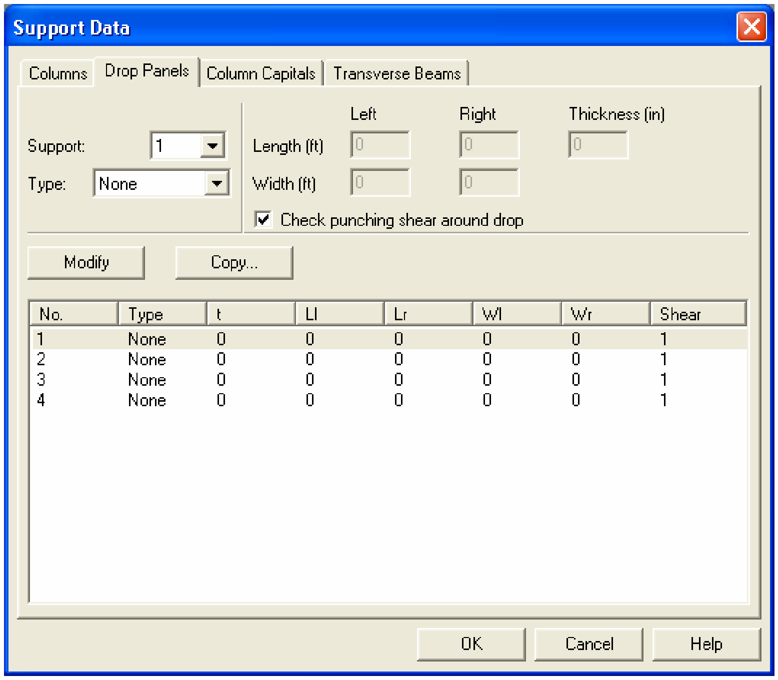


Figure 4-16 Drop Geometry Dialog Box

2. Select whether **pcaSlab** should compute the drop dimensions or the dimensions will be user specified. If **pcaSlab** is to compute the dimensions, the "Standard" option should be selected from the Type drop-down list and then only the drop depth will be available. When the "Standard drop" option is selected **pcaSlab** will calculate drop panel dimensions in accordance with ACI 318-99 Clause 13.3.7. Similar requirements contained in previous editions of the CSA A23.3 Standard have been removed from the 1994 edition. As a result, the ACI minimum specifications for drop panels are also used in CSA A23.3 runs when the "Standard Drops" option is selected. If you would like to specify drop dimensions other than those computed by **pcaSlab**, you must select "User-defined" from the Type drop-down list.
3. Enter the dimension in the direction of analysis from the column centerline to the edge of the drop left of the column (see Figure 4-17). If this is a standard drop, this dimension will not be available and the length left is set equal to the slab span length left/6 for interior columns or the left cantilever length for the first column.

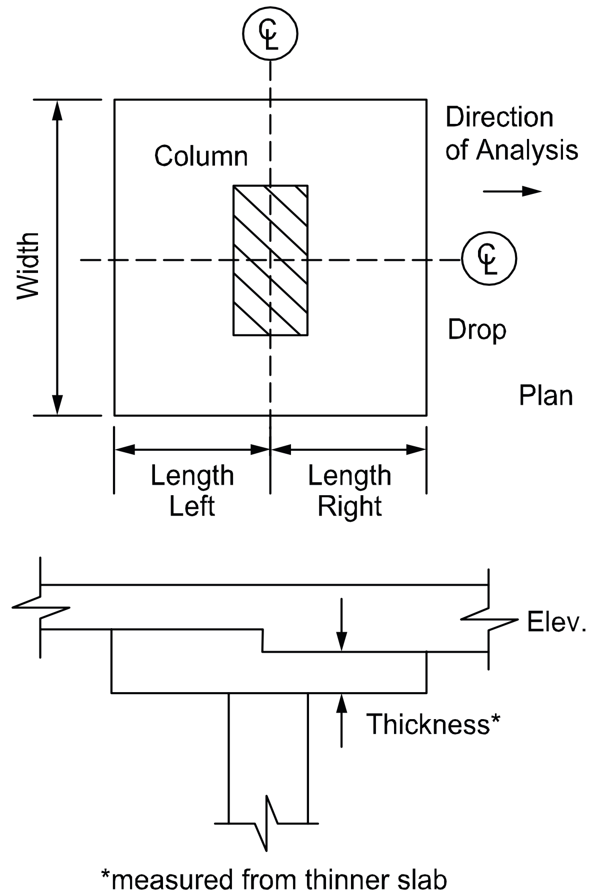


Figure 4-17 Required Drop Dimensions

4. Enter the width dimension in the transverse direction (see Figure 4-17). If this is a standard drop, this dimension will not be available and the width is set equal to slab width/3.
5. In order for **pcaSlab** to recognize drops, drop depths are required for the flat slab systems even if Standard Drop is selected. Enter the depth of the drop from the span with the smaller slab depth (see Figure 4-17). For waffle slab systems, the depth is automatically assumed to be equal to the rib depth below the slab and is not displayed. A value entered will be considered to exist below the rib depth during calculations.
6. Press the **Modify** button to update the drop geometry.
7. Repeat steps 2 through 6 until all the drop dimensions have been updated. You can use the **Copy** button as a shortcut.
8. Press **Ok** button to exit the dialog box so that **pcaSlab** will use the new drop geometry.

Defining the Column Capitals

To input column capital geometry:

1. Select the **Supports** command from the Input menu or click the  button on the tool bar. Click left mouse button on the **Column Capitals** tab to activate it. The dialog box of Figure 4-18 will appear.

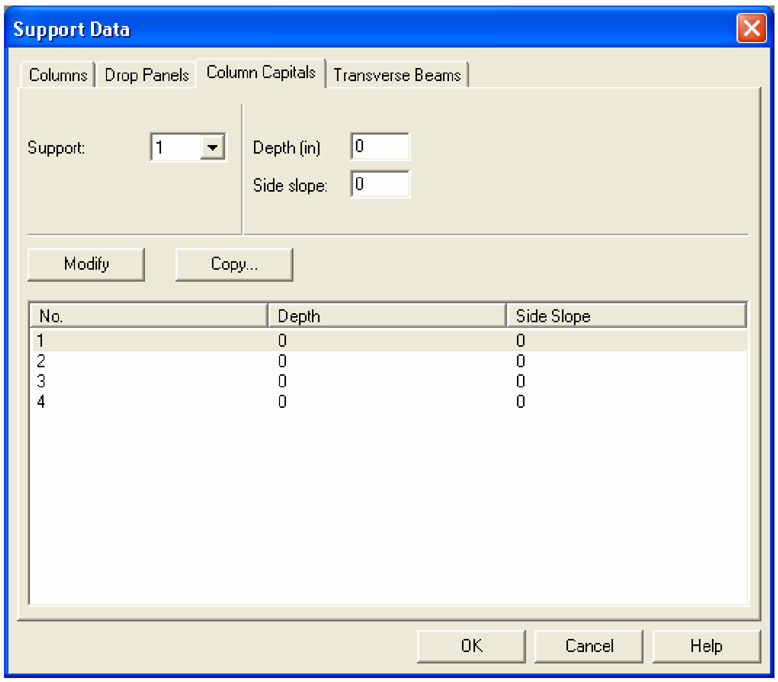


Figure 4-18 Capital Dimensions

2. Select the support number from the **Support** drop-down list.
3. Enter the capital depth which is the distance from the bottom of the soffit (slab, drop, or beam), to the bottom of the capital.
4. The Side slope is the rate of depth to extension of the capital and it must be less than 1 and smaller than 50 (see Figure 4-19).

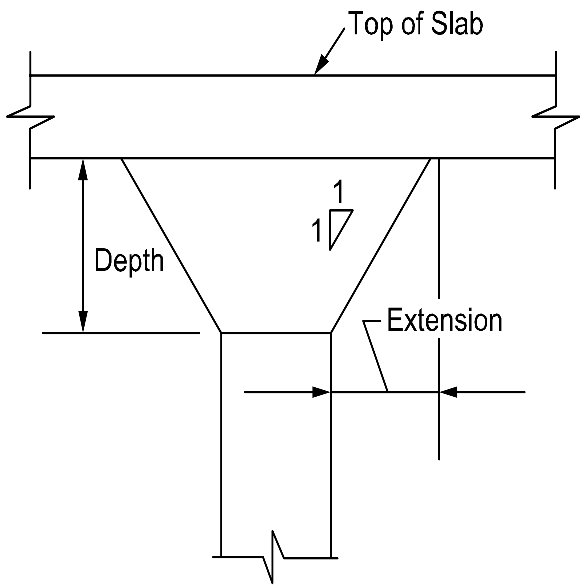


Figure 4-19 Required Capital Dimensions

5. Enter the capital extension which is the distance from the edge of the column to the end of the capital (see Figure 4-19).

Defining the Transverse Beams

The Transverse Beam command allows you to input the width, depth, and offset (eccentricity) of transverse beams at each column. This command is not available for the one-way slab systems and it is optional for the remaining systems.

To input transverse beam geometry:

1. Select the Support command from the Input menu. Select Transverse Beams tab from the Support Data dialog box. The dialog box of Figure 4-20 will appear.

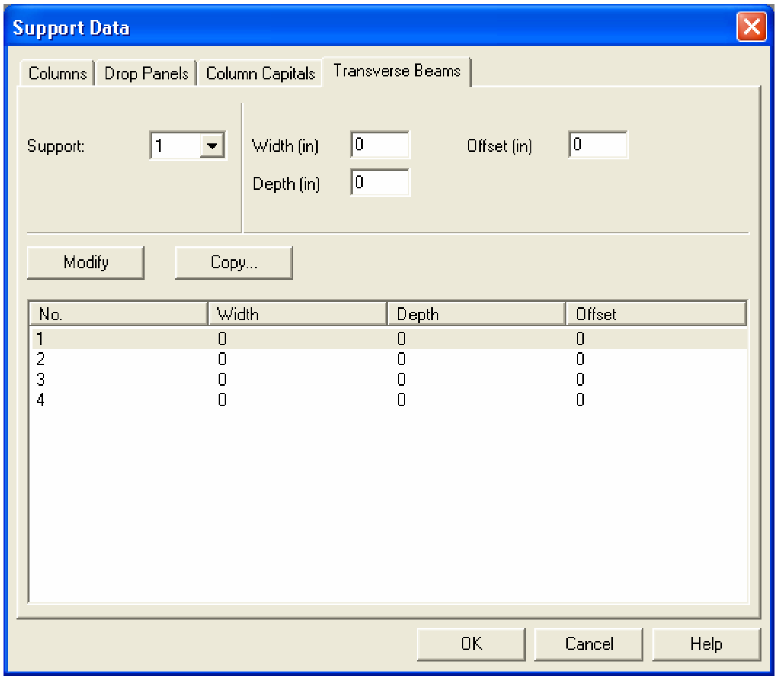


Figure 4-20 Transverse Beam Geometry dialog box

2. Enter the width of the transverse beam (see Figure 4-21)

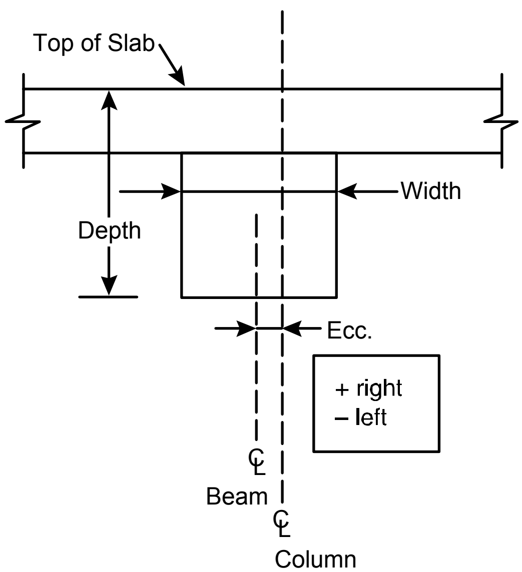


Figure 4-21 Required Transverse Beam Dimensions

3. Enter the depth of the transverse beam which is taken from the top of the slab to the bottom of the beam (see Figure 4-21)
4. Enter the eccentricity, which is measured from the joint centerline, positive to the

right, and negative to the left of the joint (See Figure 4-21).

5. Press the Modify button to update the transverse beam dimensions.
6. Repeat steps 2 through 5 until all the beams have been updated. You can use the **Copy** button as a shortcut (see "Entering the Structure Geometry" earlier in this chapter for help on the **Copy** button).
7. Press **Ok** button to exit the dialog box so that **pcaSlab** will use the new beam geometry.

Reinforcement Criteria

Defining the Reinforcement Criteria for Slabs and Ribs

In order for **pcaSlab** to select the reinforcement, you must define the slab and rib reinforcement, bar sizes, location, and minimum spacing dimensions. See "Area of Reinforcement" and "Reinforcement Selection" in Method of Solution for a discussion of the reinforcement computations.

To define reinforcement Criteria for Slabs and Ribs:

1. Select the **Reinforcement Criteria** command from the Input menu or click the 

button on the tool bar. Select **Slabs and Ribs** tab by clicking the left mouse button on the tab title. The dialog boxes of Figure 4-22 will appear.

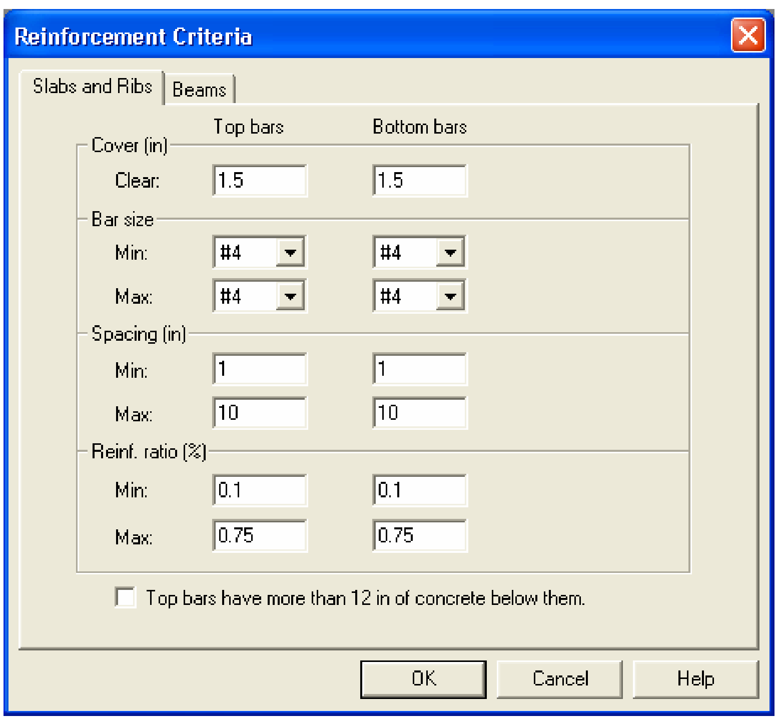


Figure 4-22 Slab and Rib Reinforcement Dialog Boxes

2. For slabs and ribs, enter the clear covers for top and bottom reinforcing bars. For the top reinforcement, this distance is from the top of the slab to the top of the top bars. For the bottom reinforcement, this distance is from the bottom of the slab to the bottom of the bottom bars (see Figure 4-23). The default value is 1.5 in. (40mm) for both input items.

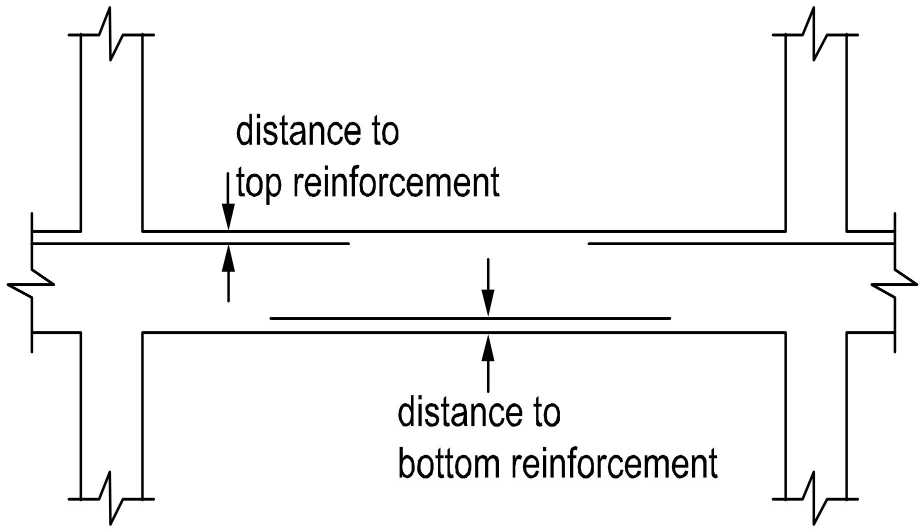


Figure 4-23 Distance to Reinforcement Edges

3. Enter the minimum bar size to start the iteration for determining flexural reinforcement.
4. Enter the maximum bar size. This number will be used as a stop in the iteration for determining flexural bars in beams.
5. Enter minimum bar spacing for slab and rib flexural reinforcement. This number should be based on aggregate size or detailing considerations. Default spacing is 6 in. (150mm) for slabs and ribs.
6. Enter maximum bar spacing for slab and rib flexural reinforcement. Default spacing is 18 in. for slabs and ribs.
7. Enter minimum Reinforcement Ratio for slab and rib flexural reinforcement. Default ratio is 0.2% for slabs and ribs. If the user specified value is smaller than 0.18, 0.18 is used by **pcaSlab**. If the user specified value is greater than 0.18, the specified value is used by **pcaSlab**.
8. Enter maximum Reinforcement Ratio for slab and rib flexural reinforcement. Default ratio is 2% for slabs and ribs.
9. If the top bars have more than 12 in. of concrete below them, check the corresponding check box.
10. Press **Ok** button to exit the dialog box and allow **pcaSlab** to use the new data.

Defining the Reinforcement Criteria for Beams

In order for **pcaSlab** to select the reinforcement, you must define the beam reinforcement, bar sizes, location, and minimum spacing dimensions. See "Area of Reinforcement" and "Reinforcement Selection" in the Method of Solution for a discussion of the reinforcement computations.

To define reinforcement criteria for beams:

1. Select the **Reinforcement Criteria** command from the Input menu or click the  button on the tool bar. Select **Beams** tab by clicking the left mouse button on the tab title. The dialog boxes of Figure 4-24 will appear.

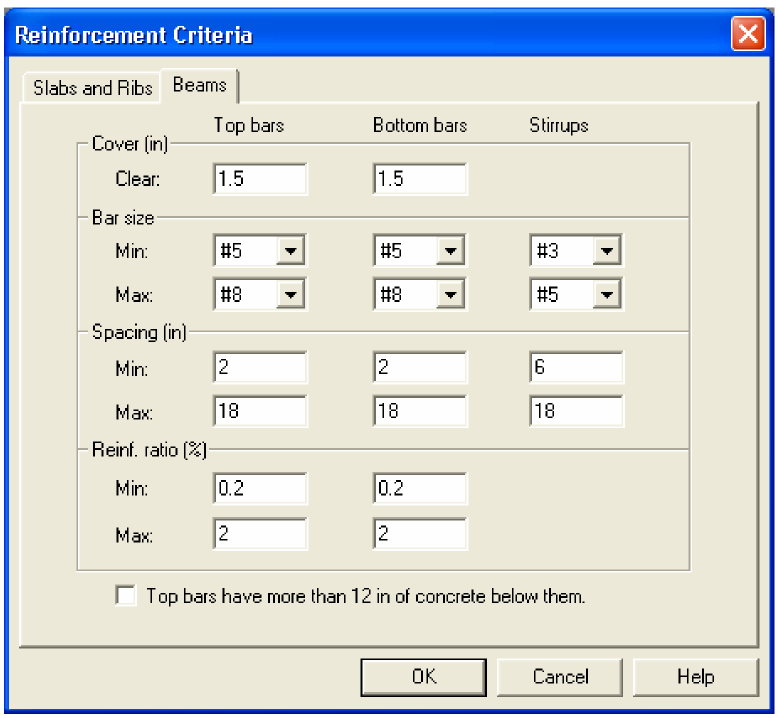


Figure 4-24 Reinforcement Dialog Boxes

2. Enter the covers for top and bottom reinforcing bars for beams. For the top reinforcement, this distance is from the top of the slab to the centroid of the top bars; and for the bottom reinforcement, this distance is from the bottom of the slab to the centroid of the bottom bars (see Figure 4-25). The default value is 1.5 in. (40mm) for both input items.

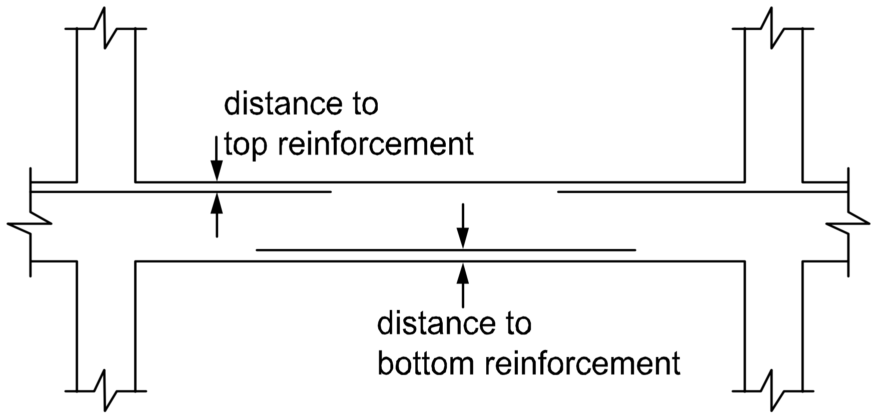


Figure 4-25 Distance to Reinforcement

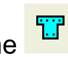
3. Enter the minimum bar size for top and bottom bars and stirrups to start the iteration for determining flexural reinforcement.
4. Enter the maximum bar size for top and bottom bars and stirrups. This number will be used as a stop in the iteration for determining flexural bars in beams.
5. Enter the minimum bar spacing for beam flexural reinforcement and stirrups. This number should be based on aggregate size or detailing considerations. The default minimum reinforcement bar spacing is 2 in. and the default stirrup spacing is 6 in.
6. Enter the maximum bar spacing for beam flexural reinforcement and stirrups. The default maximum reinforcement spacing is 18 in. and the default maximum stirrup spacing is 18 in.
7. Enter the minimum Reinforcement Ratio for beam flexural reinforcement. Default ratio is 0.2% for beams. If the user specified value is smaller than 0.2, 0.2 is used by **pcaSlab**. If the user specified value is greater than 0.2, the specified value is used by **pcaSlab**.
8. Enter the maximum Reinforcement Ratio for beam flexural reinforcement. Default ratio is 2% for slabs and ribs.
9. If the top bars have more than 12 in. of concrete below them, check the corresponding check box.
10. Press **Ok** button to exit the dialog box and allow **pcaSlab** to use the new data.

Reinforcing Bars

Defining Column Strip Bars

The reinforcing bar size, number of bars, bar length, etc. can be defined by users if the Run Mode of Investigation is selected in the General Information dialog box. This menu item is not available if Run Mode of Design is selected in the General Information dialog box.

To define column strip bars:

1. Select **Reinforcing bars** from the Input menu or click the  button on the tool bar. Select the **Column Strip Bars** tab by clicking the tab title or use the tab key on the keyboard to toggle to the tab title then select the Column Strip Bars tab using the arrow keys. The dialog boxes of Figure 4-26 will appear.

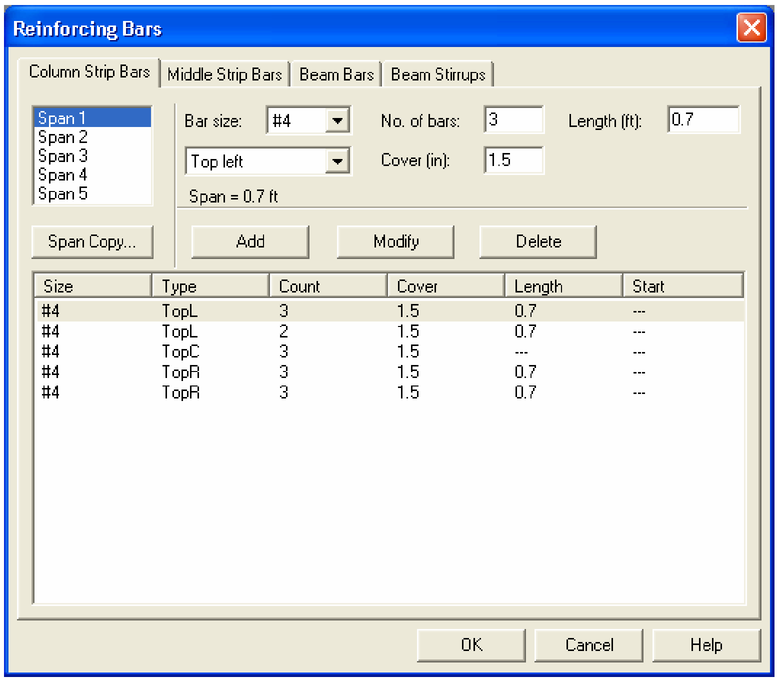


Figure 4-26 Defining Column Strip Bars

2. Select the span for which reinforcing bars will be defined from the **Span** list box on the upper left corner. The length of the selected span will be shown right above the **Add** button.
3. Select bar size from the Bar Size drop-down list.
4. Define the number of reinforcing bars in the selected span by entering the number in the **Number of Bars** input box.
5. Define the length of the reinforcing bars by entering the length in the **Length** input box. The unit of the length is foot.
6. Define the types of the reinforcing bars in the selected span by selecting from the **Type** drop-down list, which is right below the **Bar Size** drop-down list. Five types are available: Top Left, Top Right, Top Continuous, Bottom Continuous and Bottom Discontinuous.
7. Define the cover by entering the number in the **Cover** input box. The unit of cover is inch.
8. Press **Add** button to add the new data into the list box below the buttons.
9. Repeat steps 2 to 8 to define reinforcing bars for all the spans. You may use the **Span Copy** button below the **Span** list to simply copy the data of one span to other spans.

10. If the reinforcing data of a span needs to be modified, select the data from the data list box on the lower part of the dialog box then modify the data as mentioned above. Press **Modify** button when finished to update the corresponding data in the data list box.
11. To delete the reinforcing data for a span, select the data of the span from the data list box then press the **Delete** button.
12. Press **Ok** button to exit the dialog box so that **pcaSlab** will use these reinforcing bar properties.

Defining Middle Strip Bars

The reinforcing bar size, number of bars, bar length, etc., can be defined by users if the Run Mode of Investigation is selected in the General Information dialog box. This menu item is not available if Run Mode of Design is selected in the General Information dialog box.

To define middle strip bars:

1. Select **Reinforcing bars** from the Input menu or click the  button on the tool bar. Select the **Middle Strip Bars** tab by clicking the tab title or use the tab key on the keyboard to toggle to the tab title then select the Middle Strip Bars tab using the arrow keys. The dialog boxes of Figure 4-27 will appear.

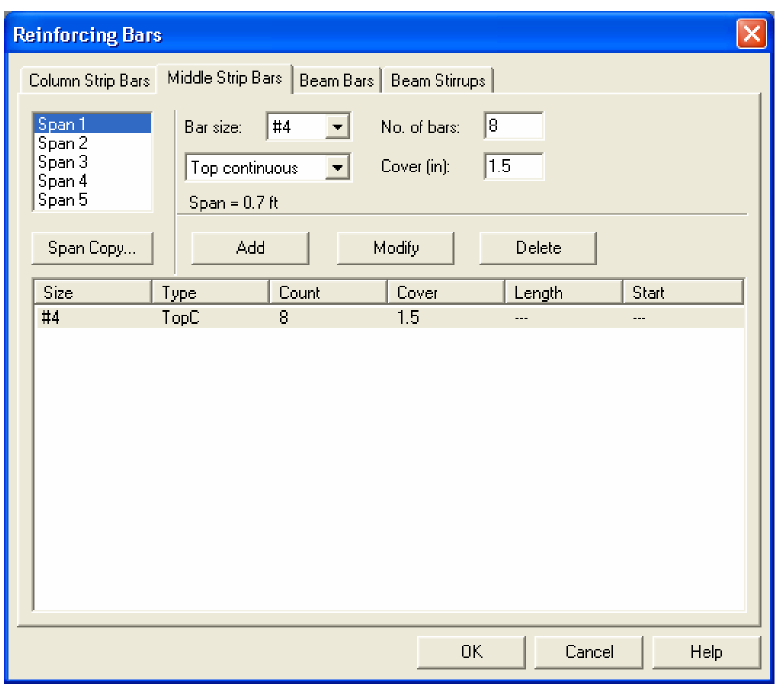


Figure 4-27 Defining Middle Strip Bars

2. Select the span for which reinforcing bars will be defined from the **Span** list box

on the upper left corner. The length of the selected span will be shown right above the **Add** button.

3. Select bar size from the Bar Size drop-down list.
4. Define the number of reinforcing bars in the selected span by entering the number in the **Number of Bars** input box.
5. Define the length of the reinforcing bars by entering the length in the **Length** input box. The unit of the length is foot.
6. Define the Types of the reinforcing bars in the selected span by selecting from the drop-down list, which is right below the **Bar Size** drop-down list. Five types are available: Top Left, Top Right, Top Continuous, Bottom Continuous and Bottom Discontinuous.
7. Define the cover by entering the number in the **Cover** input box. The unit of cover is inch.
8. Press **Add** button to add the new data into the list box below the buttons.
9. Repeat steps 2 to 8 to define reinforcing bars for all the spans. You may use the **Span Copy** button below the **Span** list to simply copy the data of one span to other spans.
10. If the reinforcing data of a span needs to be modified, select the data from the data list box on the lower part of the dialog box then modify the data as mentioned above. Press **Modify** button when finished to update the corresponding data in the data list.
11. To delete the reinforcing data for a span, select the data of the span from the data list box then press the **Delete** button.
12. Press **Ok** button to exit the dialog box so that **pcaSlab** will use these reinforcing bar properties.

Defining Beam Bars

The reinforcing bar size, number of bars, bar length, etc. can be defined by users if the Run Mode of Investigation is selected in the General Information dialog box. This menu item is not available if Run Mode of Design is selected in the General Information dialog box.

To define beam bars:

1. Select **Reinforcing bars** from the Input menu or click the button from the tool bar. Select the **Beam Bars** tab by clicking the tab title or use the tab key on the keyboard to toggle to the tab title then select the Beam Bars tab using the arrow keys. The dialog boxes of Figure 4-28 will appear.

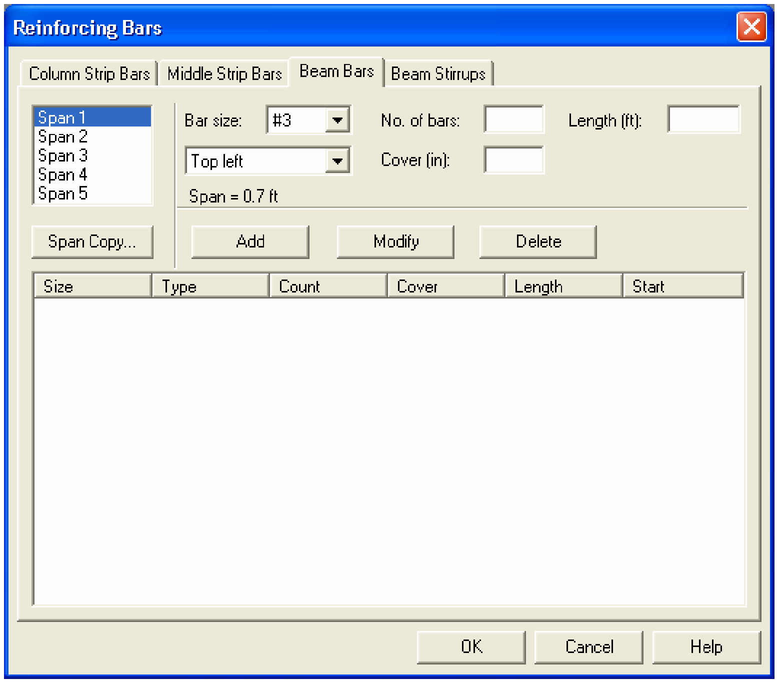
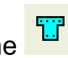


Figure 4-28 Defining Beam Bars

2. Select the span for which reinforcing bars will be defined from the **Span** list box on the upper left corner. The length of the selected span will be shown right above the **Add** button.
3. Select bar size from the **Bar Size** drop-down list.
4. Define the number of reinforcing bars in the selected span by entering the number in the **Number of Bars** input box.
5. Define the length of the reinforcing bars by entering the length in the **Length** input box. The unit of the length is foot.
6. Define the types of the reinforcing bars in the selected span by selecting from the **Type** drop-down list, which is right below the Bar Size drop-down list. Five types are available: Top Left, Top Right, Top Continuous, Bottom Continuous and Bottom Discontinuous.
7. Define the cover by entering the number in the **Cover** input box. The unit of cover is inch.
8. Press **Add** button to add the new data into the list box below the buttons.
9. Repeat steps 2 to 8 to define reinforcing bars for all the spans. You may use the **Span Copy** button below the **Span** list to simply copy the data of one span to other spans.
10. If the reinforcing data of a span needs to be modified, select the data from the data list box on the lower part of the dialog box then modify the data as mentioned above. Press **Modify** button when finished to update the corresponding data in the data list.
11. To delete the reinforcing data for a span, select the data of the span from the data list box then press the **Delete** button.
12. Press **Ok** button to exit the dialog box so that **pcaSlab** will use these reinforcing bar properties.

Defining Beam Stirrups

To define beam stirrups:

1. Select **Reinforcing bars** from the **Input** menu or click the  button from the tool bar. Select the **Beam Stirrups** tab by clicking the tab title or use the tab key on the keyboard to toggle to the tab title then select the Beam Stirrups tab using the arrow keys. The dialog boxes of Figure 4-29 will appear.

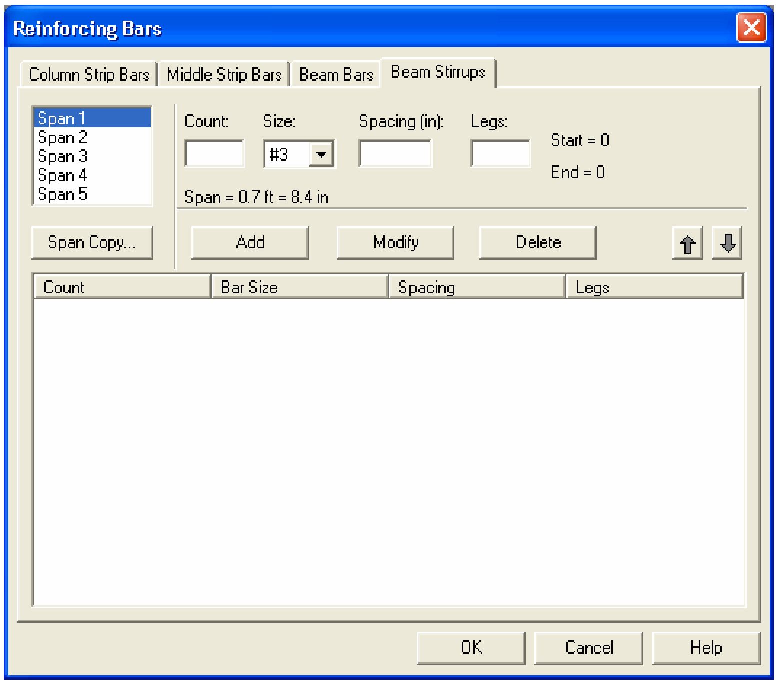


Figure 4-29 Defining Beam Stirrups

2. Select the span for which stirrups will be defined from the **Span** list box on the upper left corner. The length of the selected span will be shown right above the Add button.
3. Enter the amount of stirrups of the selected span in the **Count** input box. (see Note)
4. Select stirrup size from the **Size** drop-down list.
5. Enter the spacing of the stirrups of the selected span in the Spacing input box. The unit of spacing is inch.
6. Enter the number of legs in the **Leg** input box.
7. Press **Add** button to add the new data into the list box below the buttons.
8. Repeat steps 2 to 8 to define stirrups for all the spans. You may use the **Span Copy** button below the **Span** list to simply copy the data of one span to other spans.

9. If the stirrup data of a span needs to be modified, select the data from the data list box on the lower part of the dialog box then modify the data as mentioned above. Press **Modify** button when finished to update the corresponding data in the data list.
10. To delete the stirrup data for a span, select the data of the span from the data list box then press the **Delete** button.
11. Press **Ok** button to exit the dialog box so that **pcaSlab** will use these reinforcing bar properties.

Note:

If stirrups do not apply in a part of a span, the **Count** should be set to 0 (zero) and the **Spacing** should be the length of the part of the span where no stirrups are defined. For example, the following configuration shows stirrups in the left and right ends of a span with an empty space (46.0 in. long, no stirrups) in the middle part of the span.

Count	Bar Size	Spacing	Legs
32	#5	4.56	2
0	#5	46.0	2
32	#5	4.56	2

Load Cases

Defining Load Cases

Up to 6 load cases of dead load, live load or lateral load can be defined in the **Load Cases** dialog box. The default three load case labels (types) are SELF (dead load), Dead (dead load) and Live (live load).

To define load cases:

1. Select Load Cases command from the Input menu or click the 

button on the tool bar. Figure 4-30 will appear.

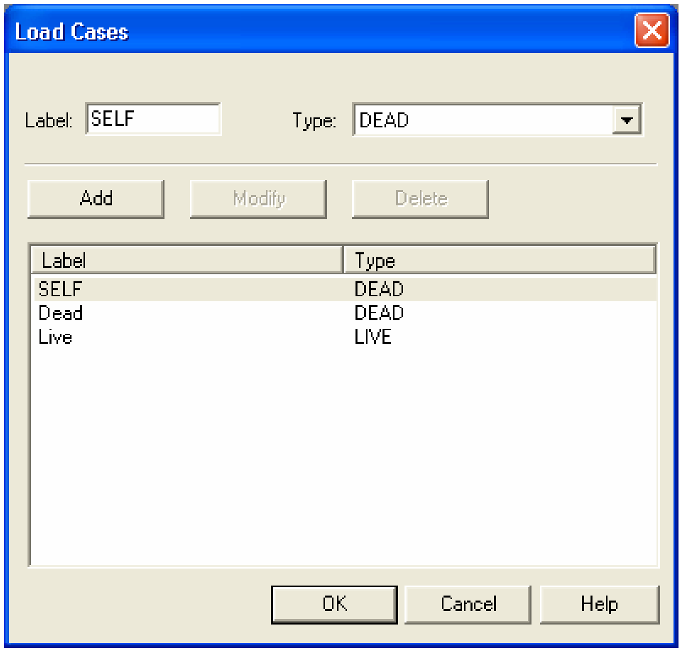


Figure 4-30 Define Load Cases

2. Enter a name for the new load case in the **Label** edit box. The name could be any character string defined by the user.
3. Select the type of the new load case from the **Type** drop-down list. The available types are Dead Load, Live Load and Lateral Load.
4. Press the **Add** button to add the new load case into the load case list box on the lower part of the dialog box.
5. Repeat steps 2 to 4 to define all the load cases. The maximum number of load cases is 6. Once the maximum number is reached, the **Add** button will be disabled.
6. To modify an existing load case, select the load case from the load case list box then change the label or type the selected load case as mentioned above and press the **Modify** button.
7. To delete an existing load case, select the load case from the load case list box then press the **Delete** button.
8. Press **Ok** button to exit the dialog box so that **pcaSlab** will use these load cases.

Note:

1. Only one case of live load can be defined.
2. Load case label must be unique for each of the load cases.

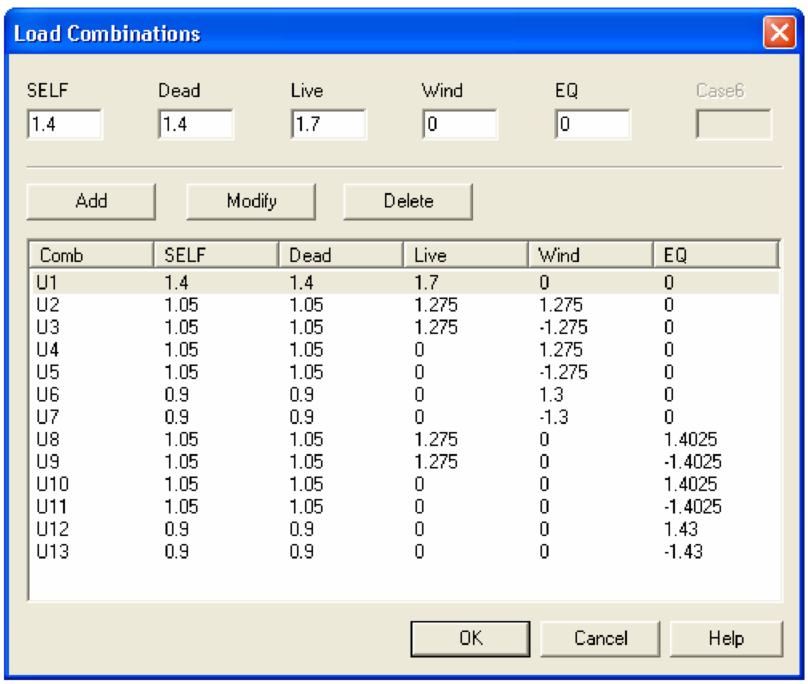
Load Combinations

Defining Load Combinations

pcaSlab allows you to change the magnification factors applied to the load cases. The default values depend on the code selected with the General Information command.

To define load factors:

1. Select **Load Combinations** from the Input menu or click the button from the tool bar. The dialog box of Figure 4-31 will appear if the ACI code was selected in the General Information dialog box.



Comb	SELF	Dead	Live	Wind	EQ
U1	1.4	1.4	1.7	0	0
U2	1.05	1.05	1.275	1.275	0
U3	1.05	1.05	1.275	-1.275	0
U4	1.05	1.05	0	1.275	0
U5	1.05	1.05	0	-1.275	0
U6	0.9	0.9	0	1.3	0
U7	0.9	0.9	0	-1.3	0
U8	1.05	1.05	1.275	0	1.4025
U9	1.05	1.05	1.275	0	-1.4025
U10	1.05	1.05	0	0	1.4025
U11	1.05	1.05	0	0	-1.4025
U12	0.9	0.9	0	0	1.43
U13	0.9	0.9	0	0	-1.43

Figure 4-31 Define Load Combinations

2. The load cases and the corresponding factors that are defined in the **Load Cases** dialog box are shown on the top of the **Load Combinations** dialog box.
3. Enter the load factors for each of the load cases in the input box below the corresponding load case label.
4. Press the **Add** button to add the combination defined above into the big list box in the lower part of the dialog box.
5. Repeat steps 2 to 4 to define all the load combinations. Up to 12 load combinations may be defined. All the combinations are indexed automatically from U1 to U12.
6. To change the factors of an existing combination, select the load combination from the load combination list box on the lower part of the dialog box then

change the factors as mentioned above. Press the **Modify** button when finished to update the data in the load combination list box.

7. To delete an existing combination, select the load combination from the load combination list box then press the **Delete** button.
8. Select **Ok** button when all the desired load factors have been modified to exit so that **pcaSlab** will use the new data.

Span Loads

About Loads

To complete your analysis of a floor system, you must specify the applied loads. **pcaSlab** computes the self weight of the floor system. There are several types of applied loads that may be entered. They are found in the Input menu. Surface loads are placed over the entire strip. Partial loads consist of uniform or trapezoidal loads, concentrated loads, and concentrated moments that may exist anywhere within the span length. Refer to Chapter 2 "Loading" for a detailed explanation of the superimposed loading on the floor system. Up to 6 load cases can be defined by users and the default load factors can be modified by users too.

The Figure 4-32 shows the input required and the following is a description of that input.

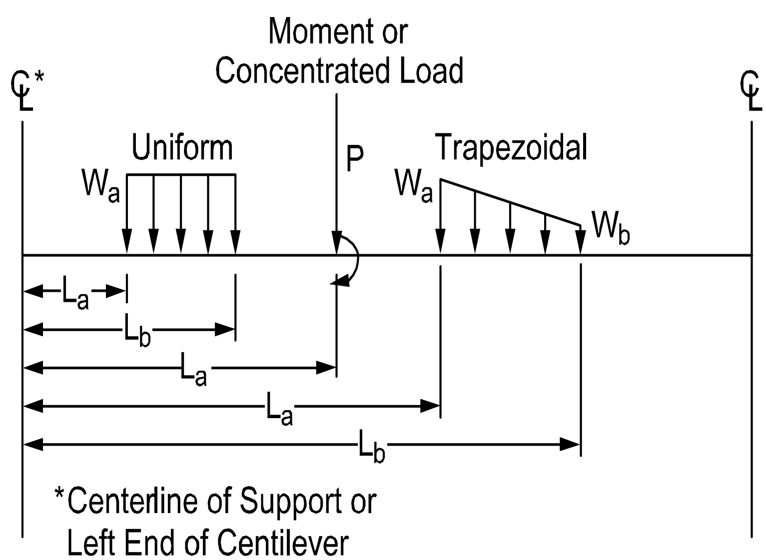


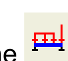
Figure 4-32 Required Partial Load Data

Wa	For uniform loads, Wa is the uniform load in units of lbs/ft (kN/m), positive Wa is downward. For trapezoidal loads, Wa is the weight at the left end in units of lbs/ft (kN/m), positive Wa is downward. For concentrated loads, Wa is the force in units of kips (kN), positive Wa is downward, and for moments, Wa is the moment in units of ft-kips (kN-m), positive Wa is clockwise.
Wb	For trapezoidal loads, Wb is the weight at the right end in units of lbs/ft (kN/m), positive Wb is downward. For all the other partial load types, Wb is not available.
La	For uniform and trapezoidal loads, La is the distance where the load begins from the centerline of the column at the left of the span, in units of ft (m). For concentrated loads and moments, La is the distance where the load exists from the centerline of the column at the left of the span in units of ft (m).
Lb	For uniform and trapezoidal loads, Lb is the distance where the load ends from the centerline of the column at the left of the span, in units of ft (m). For concentrated loads and moments, Lb is unavailable. Although the particular loading may not actually act over the entire transverse width, all line loading is converted internally by pcaSlab to act over the full width of the slab. In the design direction, partial loads given as acting over less than 1/20 of the span length will be averaged over 1/20 by the program.

Defining Area Load on Span

Area loads are uniform loads acting over the entire strip. These loads have units of lb/ft2 (kN/m2). The method for inputting area loads is the same as that used in entering the geometry.

To input area loads:

1. Select the **Span Loads** command from the Input menu or click the  button on the tool bar. Select the **Area Load** from the **Type** drop-down list on the **Span Loads** dialog box. The dialog box of Figure 4-33 will appear.

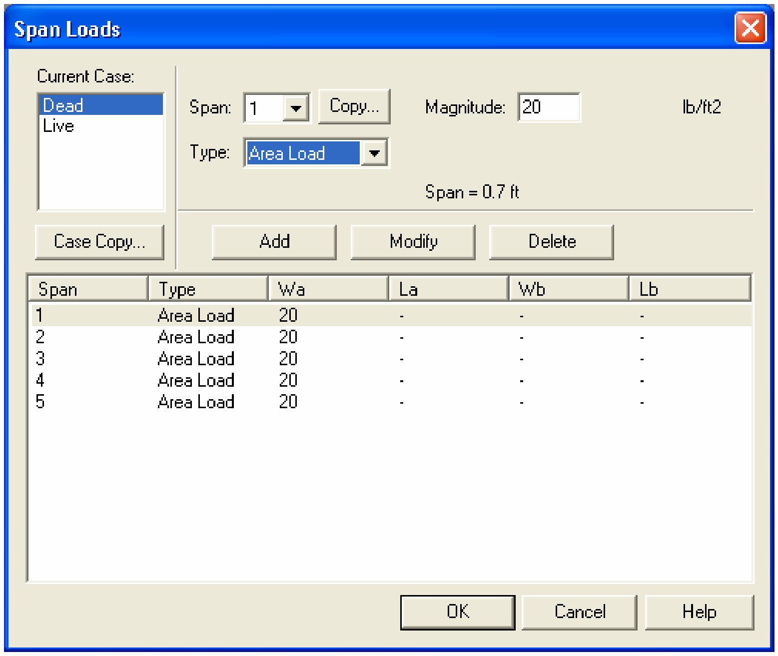


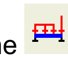
Figure 4-33 Define Area Load on Span

2. Select the load case from the **Current Case** list box on the upper left corner as shown in the dialog above.
3. Select the span on which the area loads will be applied from the Span drop-down list.
4. Enter the un-factored superimposed area load magnitude acting over the entire area of the strip in the **Magnitude** edit box. Positive surface loads act downward.
5. Press the **Add** button to update the area loads in the area load list box on the lower part of the dialog box.
6. Repeat steps 2 through 5 until the loads have been updated. You can use the **Copy** button as a shortcut.
7. To change the data of an existing area load, select the area load from the area load list box then change the magnitude as mentioned above. Press the **Modify** button when finished to update the data in the area load list box.
8. To delete an existing area load, select the load from the area load list box then press the **Delete** button.
9. Press **Ok** button to exit the dialog box so that **pcaSlab** will use the new data.

Defining Line Load on Span

You may enter uniform or trapezoidal loads that do not span from column centerline to column centerline in the direction of analysis. These loads are called line loads and are input through the Span Loads dialog of the Input menu. Partial loads are assumed to act over the entire strip width.

To input line loads:

1. Select the **Span Loads** command from the Input menu or click the  button on the tool bar. Select the **Line Load** from the **Type** drop-down list. The dialog box of Figure 4-34 will appear.

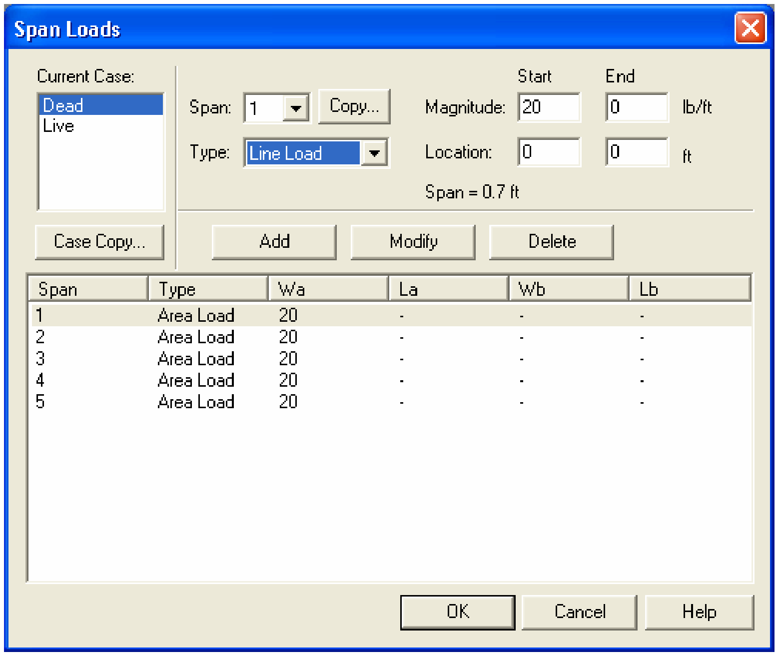


Figure 4-34 Defining Line Load on Span

2. Select the load case of the line load that will be defined from the **Current Case** list box.
3. From the **Span** drop-down list, select the span number of the span whose line loads you would like to input.
4. Define un-factored load values and their locations in the corresponding text boxes.
5. Select **Add** to add the line load defined into the line load list box.
6. Repeat steps 2 through 5 until all the line loads have been entered then press **Ok** button to exit the dialog box so that **pcaSlab** will use the new loads.

To change line loads data:

1. Select the line load you want to change from the line load list box on the lower part of the dialog box by clicking the left mouse button on the load or tabbing to the list box and using the arrow up and down keys.
2. Make your changes to the load by modifying the load type, the load magnitude, and/or location.
3. Select **Modify** to replace the old data with the new data.

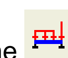
To delete line load data:

1. Select the line load you want to delete from the line load list box by clicking the left mouse button on the load or tabbing to the list box and using the arrow up and down keys.
2. Press the **Delete** button.

Defining Point Force on Span

You may enter concentrated vertical loads. These loads are input through the **Span** Load command of the Input menu. Point loads are assumed to act over the entire strip width.

To input point loads:

1. Select the **Span Loads** command from the Input menu or click the  button on the tool bar. Select the **Point Force** from the **Type** drop-down list. The dialog box of Figure 4-35 will appear.

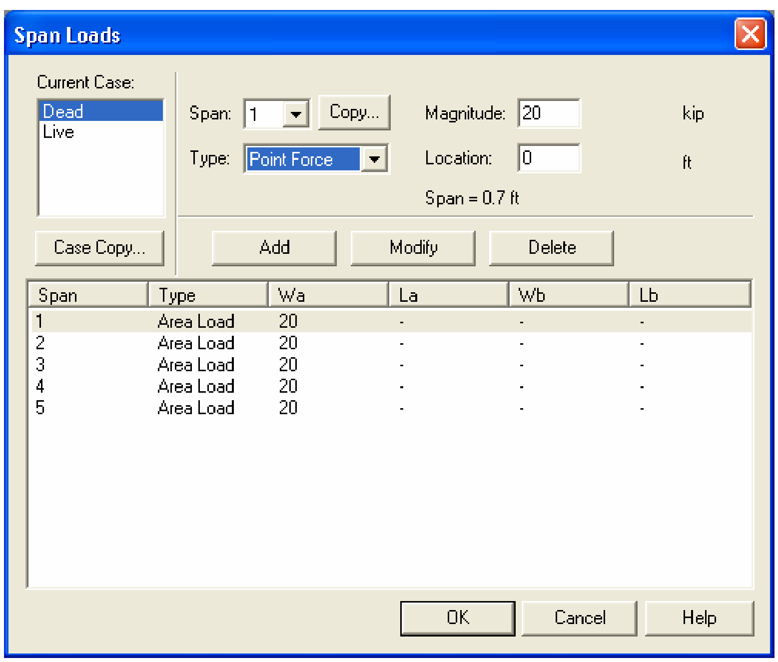


Figure 4-35 Defining Point Forces on Span

2. Select the load case of the point force that will be defined from the **Current Case** list box.
3. From the **Span** drop-down list, select the span number of the span whose point loads you would like to input.

4. Define un-factored load values and their locations in the corresponding text boxes.
5. Select **Add** to add the point load defined into the point load list box on the lower part of the dialog box.
6. Repeat steps 2 through 5 until all the point loads have been entered then press **Ok** button to exit the dialog box so that **pcaSlab** will use the new loads.

To change point load data:

1. Select the point force you want to change from the point load list box by clicking the left mouse button on the load or tabbing to the list box and using the arrow up and down keys.
2. Make your changes to the load by modifying the load type, the load magnitude, and/or location.
3. Select **Modify** to replace the old data with the new data.

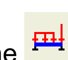
To delete point load data:

1. Select the point force you want to delete from the point load list box to the right by clicking the left mouse button on the load or tabbing to the list box and using the arrow up and down keys.
2. Press the **Delete** button.

Defining Point Moment on Span

You may enter concentrated moment. This load is input through the **Span** Load command of the Input menu. Point moments are assumed to act over the entire strip width.

To input point moments:

1. Select the **Span Loads** command from the Input menu or click the  button on the tool bar. Select the **Point Moment** from the **Type** drop-down list. The dialog box of Figure 4-36 will appear.

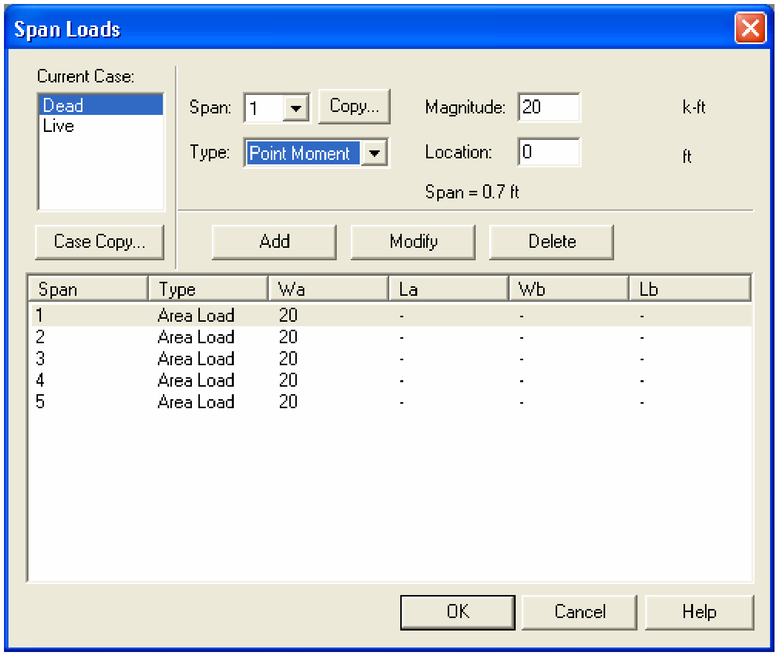


Figure 4-36 Defining Point Moments on Span

2. Select the load case of the point moment that will be defined from the **Current Case** list box.
3. From the **Span** drop-down list, select the span number of the span whose point moments you would like to input.
4. Define un-factored moment values and their locations in the corresponding text boxes.
5. Select **Add** to add the point moment defined into the point moment list box on the lower part of the dialog box.
6. Repeat steps 2 through 5 until all the point moments have been entered then press **Ok** button to exit the dialog box so that **pcaSlab** will use the new moments.

To change point load data:

1. Select the point moment you want to change from the point moment list box by clicking the left mouse button on the load or tabbing to the list box and using the arrow up and down keys.
2. Make your changes to the moment by modifying the moment type, the moment magnitude, and/or location.
3. Select **Modify** to replace the old data with the new data.

To delete point load data:

1. Select the point moment you want to delete from the point moment list box to the right by clicking the left mouse button on the load or tabbing to the list box and using the arrow up and down keys.

2. Press the **Delete** button.

Lateral Effects

Defining Lateral Effects

pcaSlab combines the gravity load analysis with a lateral load analysis. Lateral loads may be entered as joint moments depending on the type of lateral load selected in the General Information dialog box. These moments must be obtained by a frame analysis. The joint moments are combined with the gravity load moments to produce load patterns 5 through 8.

To input lateral load moments:

1. Define at least one lateral load case from the Load Cases dialog box.
2. Select **Lateral Effects** from the Input menu. If no lateral case is defined this command is disabled. Once selected, Figure 4-37 will appear.

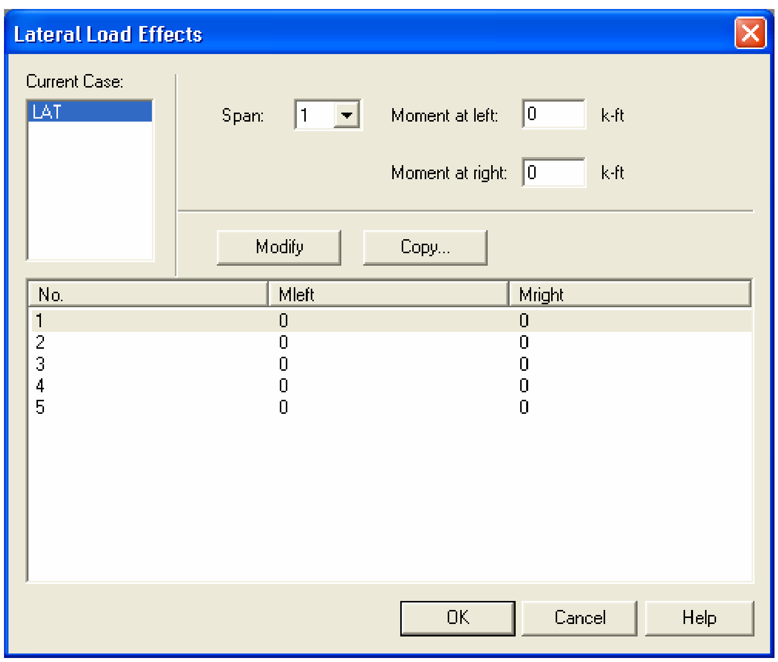


Figure 4-37 Lateral Moment Dialog Box

3. Select load cases from the **Current Case** list box. At least one lateral load case must be defined in the **Load Cases** dialog box before you can see the load case in the **Current Case** list box.
4. Select span number on which lateral loads will be defined from the **Span** drop-down list.
5. Enter the moments at the left end and right end of the span in the **Moment at**

Left and Moment at Right input boxes, respectively.

6. Press the **Add** button to add the lateral load defined above into the lateral load list box in the lower part of the dialog box.
7. Repeat steps 2 through 5 to define all the lateral loads.
8. To change an existing lateral load, select the load from the lateral load list box then change the moments as mentioned above. Press the **Modify** button when finished to update the data in the lateral load list box.
9. To delete an existing lateral load, select the load from the lateral load list box then press the **Delete** button.
10. Select **Ok** button when all the desired lateral loads have been modified to exit so that **pcaSlab** will use the new data.

Executing the Calculations (menu Solve)

Solve Options

The **Solve Options** command allows you to specify the live load pattern ratio, critical section of punching shear, and the section to calculate deflection as shown in Figure 4-38. To take effect, the design must be performed after the Condense/Expand Output command is executed.

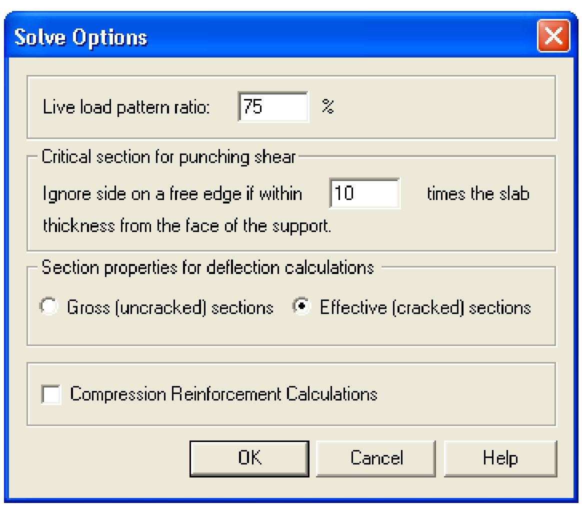
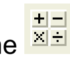


Figure 4-38 Solve Options dialog box

Execute

The **Execute** command starts the design portion of **pcaSlab** after you have finished inputting all the data.

To design the system:

Select the **Execute** command from the Solve menu or click the  button on the tool bar. If any data required to analyze and design the system has not been input prior to executing this command, **pcaSlab** will display an "Invalid model!" message. You must complete the data before execution.

An analysis status window shows the current state of the execution as shown in Figure 4-39. If the state of each of the computations is **OK**, the analysis is successful. If any error is encountered **ERROR** will be shown and the computation is terminated.

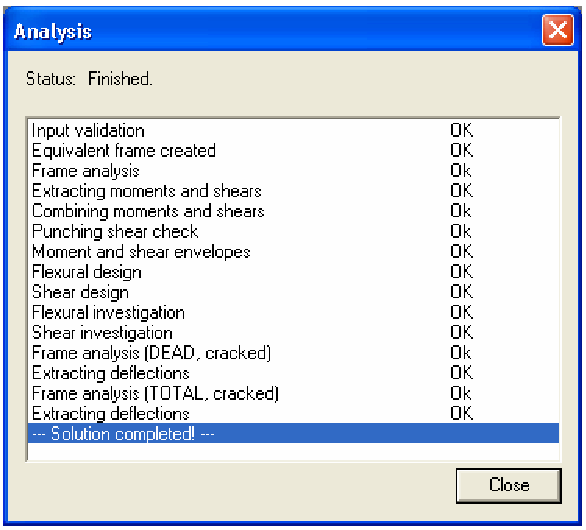


Figure 4-39 Analysis Status Window

View Program Output (menu View)

Change View Options

Zooming in on the Floor System

In order to view the floor system in greater detail, **pcaSlab** allows you to magnify a portion of the floor system for closer analysis.

Using zoom with a magnifier:

1. Select **Zoom** from the **View** menu then a sub menu appears beside the **View** menu.
2. Select **Window** from the sub menu. Notice that the **Window** command is

checked and the cursor is changed to a magnifier. The other way to magnify window is to click the  button on the tool bar.

3. Move the cross cursor to the upper left corner of the portion of the system you want to enlarge.
4. Press and hold down the left mouse button while dragging the cursor to the lower right portion of the system enclosing the desired area within the dashed box.
5. Release the mouse to enlarge that portion.

Using zoom in and zoom out command:

1. Select **Zoom** from the **View** menu and a sub menu appears.
2. Select the **In(2x)** menu command from the sub menu. The current active window will be magnified by two times. The other way to do this is to click the  button on the tool bar.
3. Select the **Out(0.5x)** menu command from the sub menu. The current active window will be reduced by two times. The other way to do this is to click the  button on the tool bar.

Note:

1. To return to original (default) zoom, select **Restore** from the **View** menu, or select  from tool bar.

2. To move model in a view window, use **Pan** from the **View** menu, or select  from tool bar.

Change the Isometric View Angle

You can modify the X and Z angles at which the floor system is displayed in the isometric view window. By default, the floor system is viewed at -45° about the X axis and 45° about the Z axis. The right hand rule is used to determine the angle of rotation about the X and Z axes and the X axis is always rotated first.

To change angles:

1. Select **Change View Angle** from the **View** menu. The dialog box of Figure 4-40 appears displaying the currently set angles.

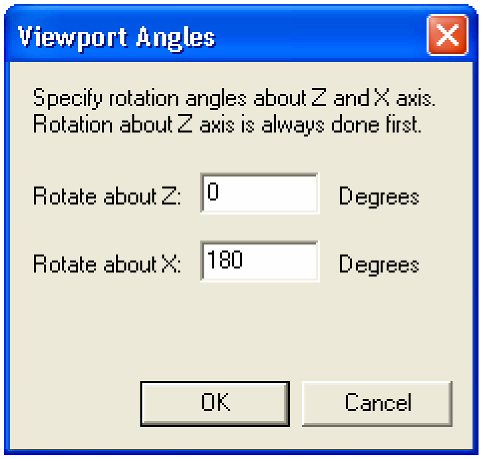


Figure 4-40 Isometric View dialog box

2. Change the angles of rotation about the X and Z axes. The X axis is rotated first then the Z axis is rotated. The floor system is rotated about the axis using the right hand rule.
3. Select **Ok** button to accept the new angles and to modify the isometric view.

Note:

A more convenient way to change the view angles instantly without entering angles in the dialog box is to use the keyboard shortcut **Ctrl + Arrow keys**. For example, press **Ctrl +** ← and **Ctrl +** → to rotate the floor system around Z axis and press **Ctrl +** ↑ or **Ctrl +** ↓ to rotate around X axis.

Viewing the Specific Member Type

In order to see the floor system members in greater detail, you can select specific member types to view by enabling and disabling them.

To select member type:

1. Select the **View Options** command from the **View** menu. The dialog box of Figure 4-41 will appear showing the currently displayed member types. Depending on the floor system, some of the member types will be shaded gray and unavailable.

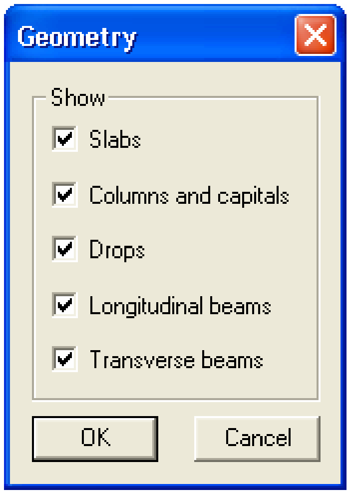


Figure 4-41 View Options Dialog Box

2. To hide a member type from a view of the floor system you must remove the ✓ sign from the check box. Click the left mouse button on the check box of the member type you would like to hide. To view a previously hidden member type, click the left mouse button on the check box to add the ✓ sign to the check box.
3. Select **Ok** button to view only the selected member types.

Plan View

To switch to plan view:

1. Select the **Plan View** command from the **View** menu or click  button on the tool bar. The dialog box of Figure 4-42 will appear.

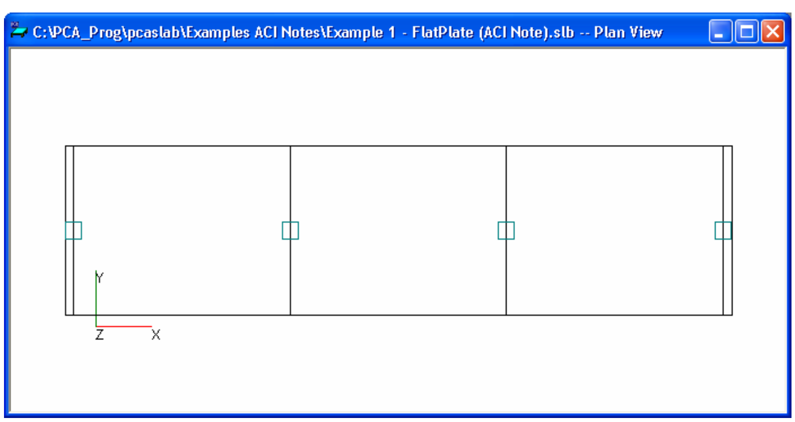


Figure 4-42 Plan View Window

2. Right click on the view window to show the pop-up menu. You may save the current view window as bitmap (BMP) file or metafile (EMF) on the Windows clipboard by selecting **Copy Bitmap** or **Copy Metafile**, respectively. You may also select the commands from the pop-up menu to zoom in or zoom out the view window and preview it before printing it out.

Figure 4-43 Pop-up Menu of View Window

3. Select the **Options** command from the pop-up menu, and then you may decide what geometry members need to be shown on the view window as shown in Figure 4-44. To hide a member type from a view of the floor system you must remove the ✓ sign from the check box. Click the left mouse button on the check box of the member type you would like to hide. To view a previously hidden member type, click the left mouse button on the check box to add the ✓ sign to the check box.

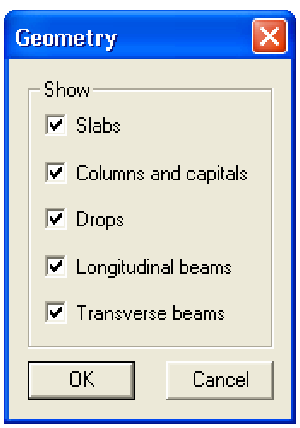
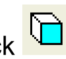


Figure 4-44 Geometry Options of Plan View

4. Select **Ok** button to view only the selected member types.

Elevated View

To switch to elevated view:

1. Select the **Elevated View** command from the **View** menu or click  button from the tool bar. The dialog box of Figure 4-45 will appear.

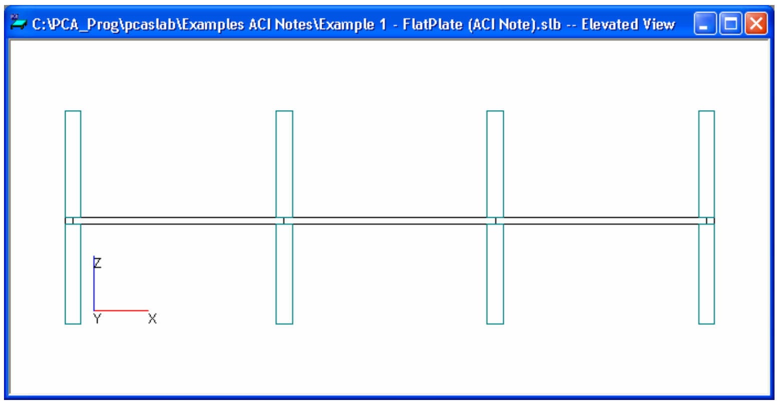


Figure 4-45 Elevated View Window

2. Right click on the view window to show the pop-up menu. You may save the current view window as bitmap (BMP) file or metafile (EMF) onto the Windows clipboard by selecting **Copy Bitmap** or **Copy Metafile**, respectively. You may also select the commands from the pop-up menu to zoom in or zoom out the view window and preview it before printing it out.

Figure 4-46 Pop-up Menu of View Window

3. Select the **Options** command from the pop-up menu, and then you may decide what geometry members need to be shown on the view window as shown in Figure 4-47. To hide a member type from a view of the floor system you must remove the ✓ sign from the check box. Click the left mouse button on the check box of the member type you would like to hide. To view a previously hidden member type, click the left mouse button on the check box to add the ✓ sign to the check box.

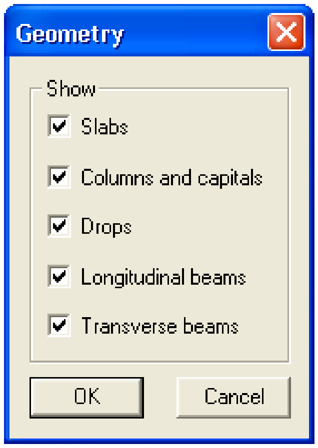


Figure 4-47 Geometry Options of Elevated View

4. Select **Ok** button to view only the selected member types.

Side View

To switch to side view:

1. Select the **Elevated View** command from the **View** menu or click  button from the tool bar. The dialog box of Figure 4-48 will appear.

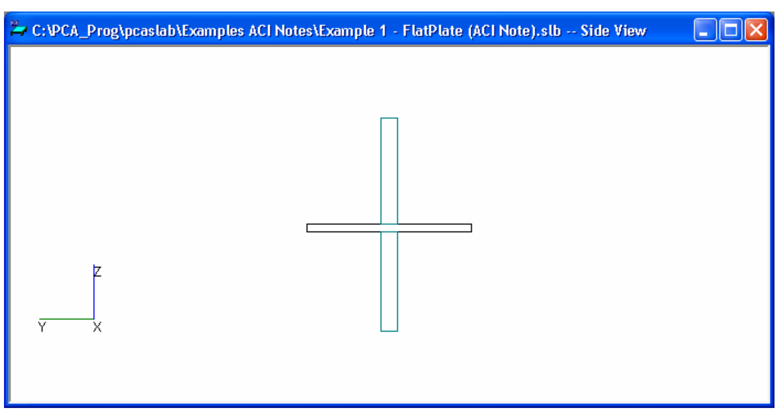


Figure 4-48 Side View Window

2. Right click on the view window to show the pop-up menu. You may save the current view window as bitmap (BMP) file or metafile (EMF) onto the Windows clipboard by selecting **Copy Bitmap** or **Copy Metafile**, respectively. You may also select the commands from the pop-up menu to zoom in or zoom out the

view window and preview it before printing it out.

Figure 4-49 Pop-up Menu of View Window

3. Select the **Options** command from the pop-up menu, and then you may decide what geometry members need to be shown on the view window as shown in Figure 4-50. To hide a member type from a view of the floor system you must remove the ✓ sign from the check box. Click the left mouse button on the check box of the member type you would like to hide. To view a previously hidden member type, click the left mouse button on the check box to add the ✓ sign to the check box.

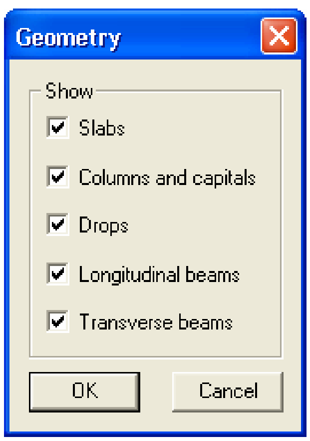


Figure 4-50 Geometry Options of Side View

4. Select **Ok** button to view only the selected member types.

Isometric View

To switch to isometric view:

1. Select the **Isometric View** command from the **View** menu or click  button on the tool bar. The dialog box of Figure 4-51 will appear.

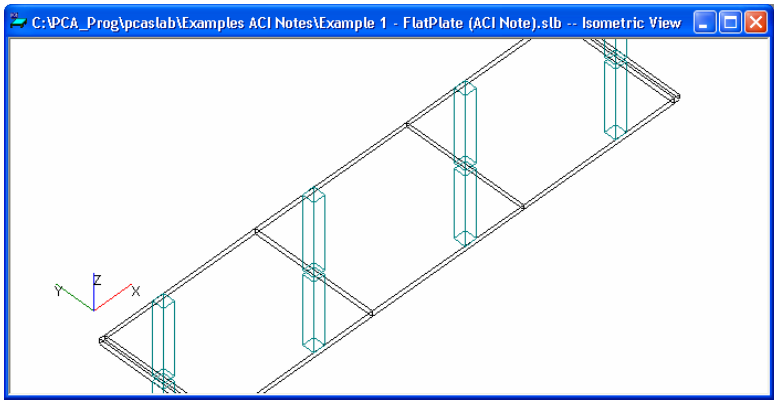


Figure 4-51 Isometric View Window

2. Right click on the view window to show the pop-up menu. You may save the current view window as bitmap (BMP) file or metafile (EMF) onto the Windows clipboard by selecting **Copy Bitmap** or **Copy Metafile**, respectively. You may also select the commands from the pop-up menu to zoom in or zoom out the view window and preview it before printing it out.

Figure 4-52 Pop-up Menu of View Window

3. Select the **Options** command from the pop-up menu, and then you may decide what geometry members need to be shown on the view window as shown in Figure 4-53. To hide a member type from a view of the floor system you must remove the ✓ sign from the check box. Click the left mouse button on the check box of the member type you would like to hide. To view a previously hidden member type, click the left mouse button on the check box to add the ✓ sign to the check box.

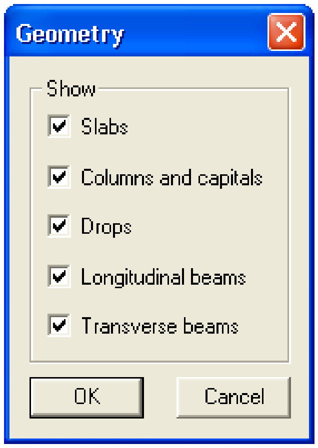
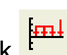


Figure 4-53 Geometry Options of Isometric View

4. Select **Ok** button to view only the selected member types.

Loads

To show the loads:

1. Select the **Loads** command from the **View** menu or click  button from the tool bar. The dialog box of Figure 4-54 will appear.

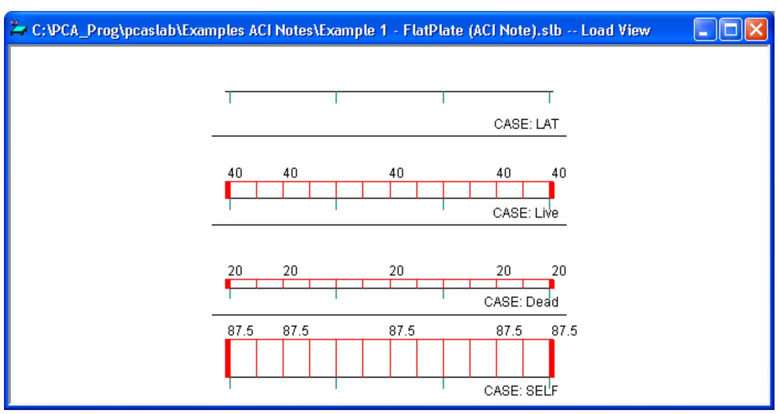


Figure 4-54 Loads View Window

2. Right click on the view window to show the pop-up menu. You may save the current view window as bitmap (BMP) file or metafile (EMF) onto the Windows clipboard by selecting **Copy Bitmap** or **Copy Metafile**, respectively. You may also select the commands from the pop-up menu to zoom in or zoom out the view window and preview it before printing it out.

Figure 4-55 Pop-up menu of View Window

3. Select the **Options** command from the pop-up menu, and then you may decide what kind of loads need to be shown on the view window as shown in Figure 4-56. To hide a load case or type you must remove the ✓ sign from the check box. Click the left mouse button on the check box of the member type you would like to hide. To view a previously hidden load type, click the left mouse button on the check box to add the ✓ sign to the check box.

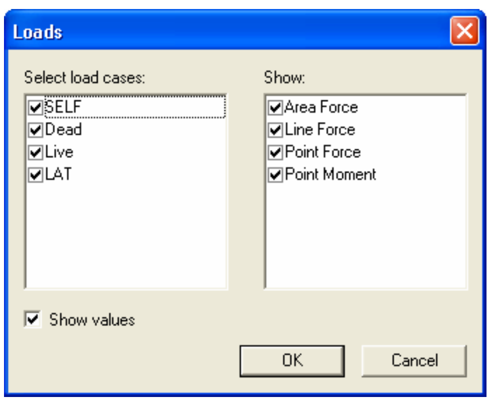


Figure 4-56 Loads Options

4. If you want to hide the load values, remove the ✓ sign before **Show values** check box.
5. Select **Ok** button to view only the selected member types.

View Results Report

Viewing Results Report

Once the analysis and/or design is performed, you can view the results, shear and moment diagrams, and deflected shapes. This section provides procedures performing these functions.

To view the analysis and design results:

1. Select **Report** command from the **Solve** menu or click the  button from the tool bar. A dialog box similar to Figure 4-57 will appear.

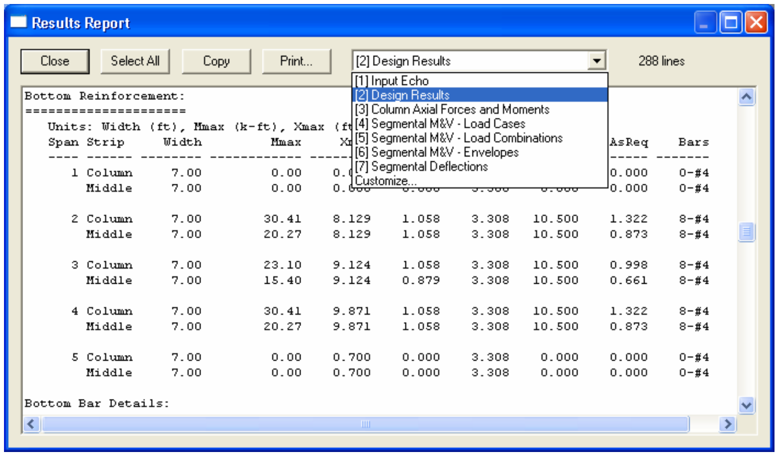


Figure 4-57 View and Print Results Dialog Box

2. Select the results you want to view from the drop-down list. The contents of the results box will be changed based on your selection. The **Customize** option allows for more than one result selection to be viewed in the results report.
3. Press the **Copy** button to copy the current results into the Windows clipboard. Then you may use **Ctrl + V** to paste the contents of the clipboard to any other editors such as Word or Notepad.
4. Press the **Close** button to close the result dialog box.

Saving Results Report

Analysis and design results are automatically saved to a text file named "projectname.out" in the same folder as the project data file (.slb file). The contents of the .out file depends on which option is selected from the drop-down list on the Results Report dialog box.

For example, if the project name is "Example 1 - FlatPlate (ACI Note)" and Input Echo is selected from the Results Report dialog box as shown in Figure 4-58, a text file named "Example 1 - FlatPlate (ACI Note).out" is generated automatically in the same directory as "Example 1 - FlatPlate (ACI Note).slb".

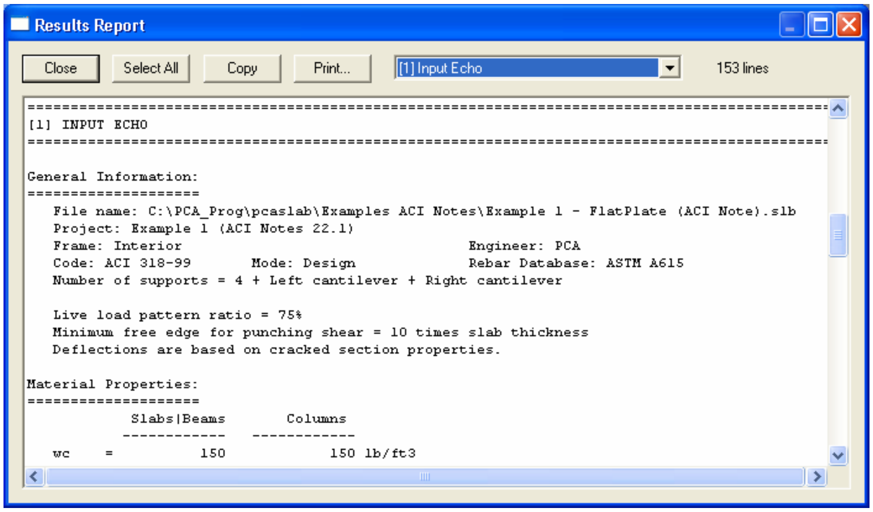


Figure 4-58 Input Echo is selected from Results Report dialog box

The file "Example 1 - FlatPlate (ACI Note).out" is a pure text file and can be opened, edited and printed using Word or Wordpad as shown in Figure 4-59.

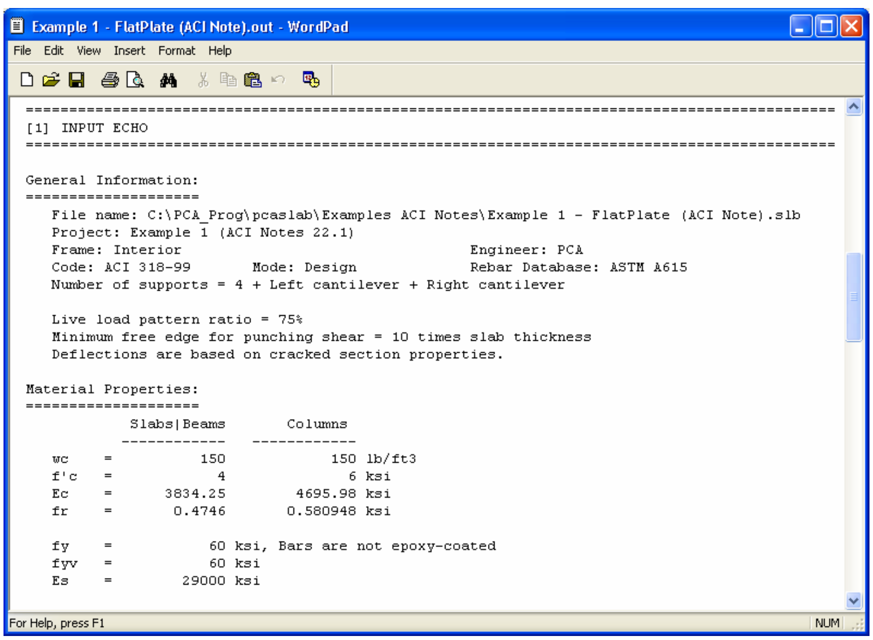


Figure 4-59 Output file is opened using Wordpad

Note:

Once another option is selected from the Results Report dialog box, the contents of the .out file is changed based on the selection.

Output files (.out file) may not be in proper format if opened using Microsoft Notepad. Please use Word or Wordpad.

View Graphical Results

Viewing the Shear and Moment Diagrams

Once the design has been performed, you may view the shear and moment diagrams for any span at any available loading pattern. The shear and moment window will be split in half horizontally. The shear diagram will occupy the upper half and the moment diagram will occupy the lower half where each diagram will be scaled to fill the entire half of the window.

To view shears and moments diagram:

1. Select the **Shears and Moments** command from the View menu or click the  button on the tool bar. A view window similar to Figure 4-60 will appear. The upper half view window shows the shear diagram and the lower half the view window shows the moment diagram.

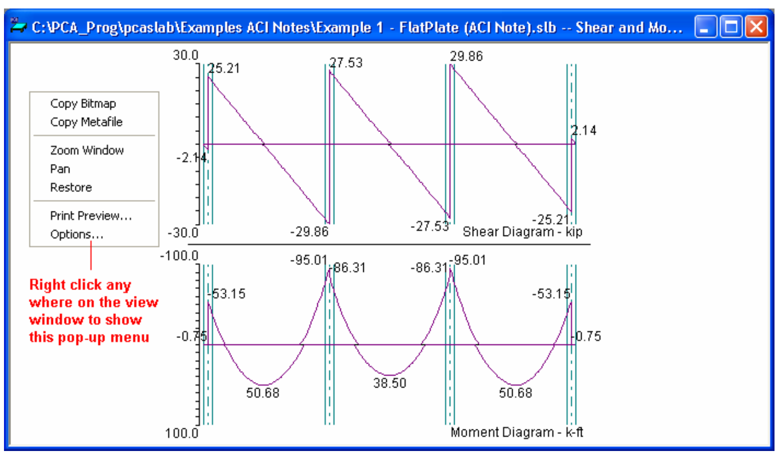


Figure 4-60 View Shear and Moment Diagram

2. The current coordinate values can be captured based on the position of the mouse curser. The status bar in Figure 4-61 shows the name of the diagram, two coordinate values of the current mouse curser position, and the current design code used in the project.

Figure 4-61 Coordinate Value Shown in Status Bar

3. Click the right mouse button anywhere on the view window to show the pop-up menu. You may **Restore, Zoom**, **Pan** or **Print View** directly by selecting

commands from this pop-up menu.

4. Click the **Options** command from the pop-up menu to change the span for which the shear and moment diagram will be shown. Figure 4-62 shows the Options dialog box. Select a span number from the **Show diagram for** drop-down list and select a load Envelope or combination from the **Select load combinations** check list box. Press the **OK** button to close the dialog box and redraw the view windows.

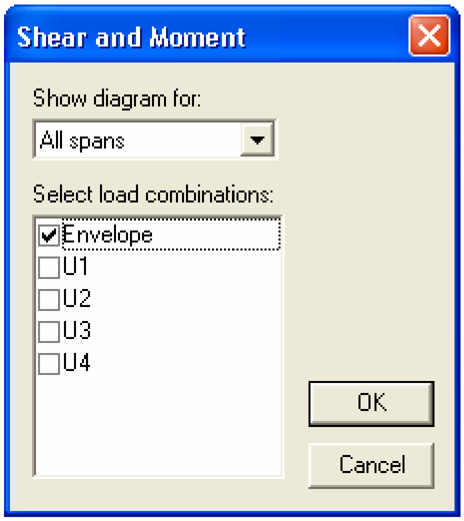


Figure 4-62 View Shear and Moment Diagram Options

Note:

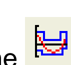
The pop-up menu can be accessed from each of the view windows of **pcaSlab**.

The **Restore** command cannot be executed in the pop-up menu until the **Zoom** or **Pan** command is stopped by pressing the **Esc** key. Without stopping the **Zoom/Pan** command, one can only restores a view using the **Restore** command in **View** menu or  button in the tool bar. This occurs on all **pcaSlab** view windows.

Viewing the Moment Capacity

Once the design has been performed, you may view the moment capacity diagrams for any span at any available loading pattern. The moment capacity window will be split in half horizontally. The middle strip moment capacity will occupy the upper half and the column strip moment capacity will occupy the lower half where each diagram will be scaled to fill the entire half of the window.

To view moment capacity:

1. Select the Moment Capacity command from the View menu or click the 

button on the tool bar. A view window similar to Figure 4-63 will appear.

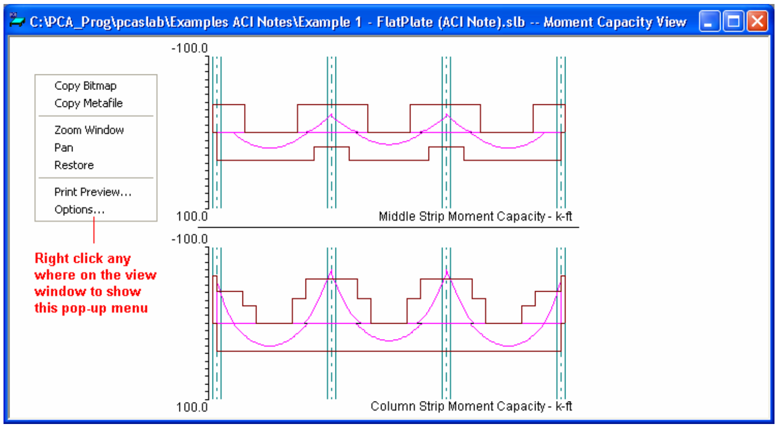


Figure 4-63 View Moment Capacity Diagram

2. Right click on the **Moment Capacity** view window and select **Options** command from the pop-up menu. A dialog box similar to Figure 4-64 will appear.

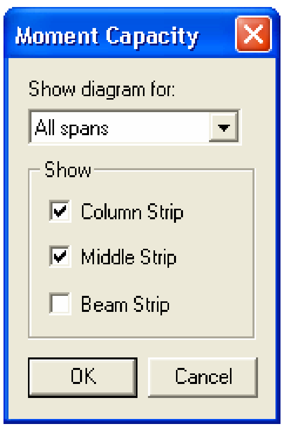


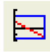
Figure 4-64 View Moment Capacity Option bialog box

3. The current coordinate values can be captured based on the position of the mouse curser. The status bar shows the name of the diagram, two coordinate values of the current mouse curser position, and the current design code used in the project.
4. Click the right mouse button anywhere on the view window to show the pop-up menu. You may **Restore**, **Zoom**, **Pan** or **Print View** directly by selecting commands from this pop-up menu.
5. Click the **Options** command from the pop-up menu to change the span for which the moment capacity will be shown. Select which span will be shown in the **Show diagram for** drop-down list. Select which part of the selected span will be shown in the **Show** frame box.
6. Press the **OK** button to close the dialog box and redraw the view windows.

Viewing the Shear Capacity

Once the design has been performed, you may view the slab shear capacity diagrams for any span at any available loading pattern.

To view shear capacity:

1. Select the Shear Capacity command from the View menu or click the  button on the tool bar. A view window similar to Figure 4-65 will appear.

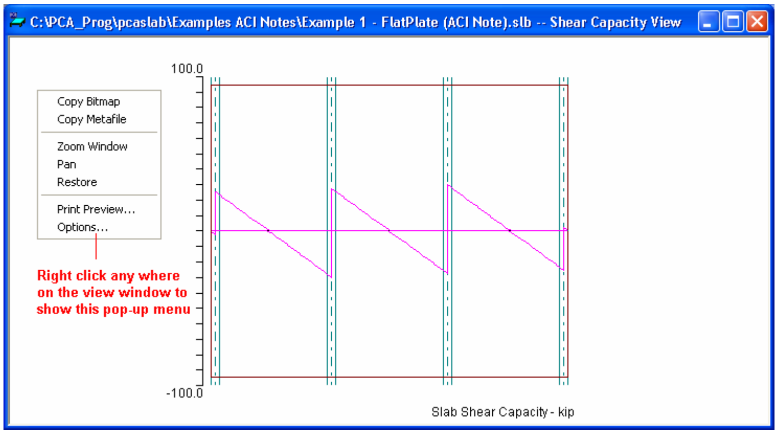


Figure 4-65 View Shear Capacity

2. Right click on the **Shear Capacity** view window and select **Options** command from the pop-up menu. A dialog box similar to Figure 4-66 will appear.

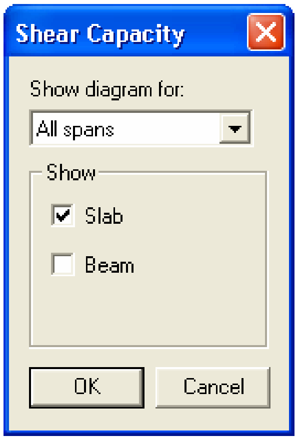


Figure 4-66 View Shear Capacity Option dialog box

3. The current coordinate values can be captured based on the position of the

mouse curser. The status bar shows the name of the diagram, two coordinate values of the current mouse curser position, and the current design code used in the project.

4. Click the right mouse button anywhere on the view window to show the pop-up menu. You may **Restore**, **Zoom**, **Pan** or **Print View** directly by selecting commands from this pop-up menu.
5. Click the **Options** command from the pop-up menu to change the span for which the shear capacity will be shown. Select which span will be shown in the **Show diagram for** drop-down list. Select which part of the selected span will be shown in the **Show** frame box.
6. Press the **OK** button to close the dialog box and redraw the view windows.

Viewing the Reinforcement

Once the design has been performed, you may view the reinforcement diagrams for any span at any available loading pattern. The reinforcement window will be split in half horizontally. The middle strip reinforcement will occupy the upper half, and the column strip reinforcement will occupy the lower half where each diagram will be scaled to fill the entire half of the window.

To view reinforcement:

1. Select the Reinforcement command from the View menu or click the  button on the tool bar. A view window similar to Figure 4-67 will appear.

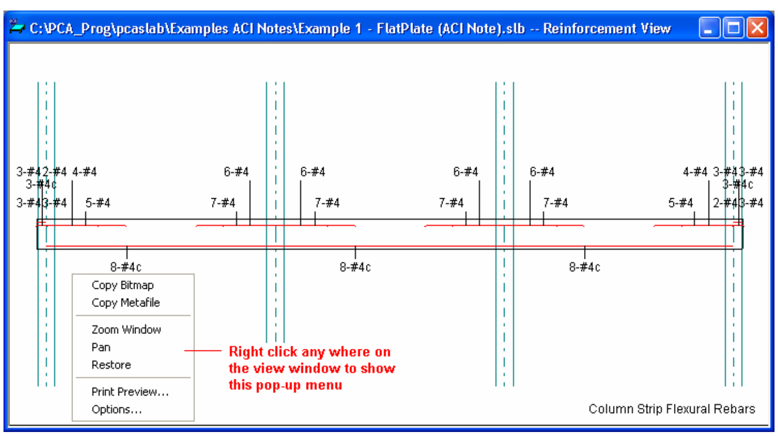


Figure 4-67 View Reinforcement

2. Right click on the **Reinforcements** view window and select **Options** command from the pop-up menu. A dialog box similar to Figure 4-68 will appear.

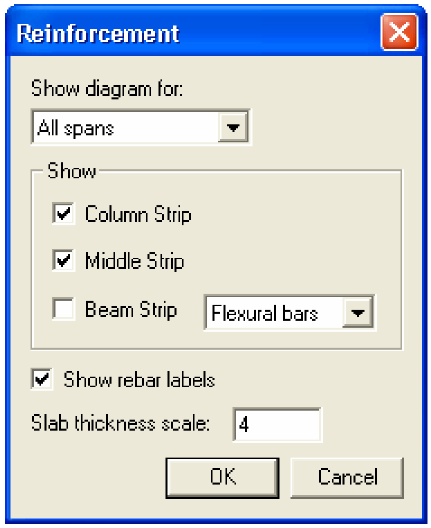


Figure 4-68 View Reinforcement Options Dialog Box

3. The current coordinate values can be captured based on the position of the mouse curser. The status bar shows the name of the diagram, coordinate value of the current mouse curser position in the design direction, and the current design code used in the project.
4. Click the right mouse button anywhere on the view window to show the pop-up menu. You may **Restore**, **Zoom**, **Pan** or **Print View** directly by selecting commands from this pop-up menu.
5. Click the **Options** command from the pop-up menu to change the span for which the shear capacity will be shown. Select the span you want to show from the **Show diagram for** drop-down list. Select which part of the selected span will be shown from the **Show** frame box. Checking the **Show rebar labels** will show labels beside each rebar on the view.
6. Press the **OK** button to close the dialog box and redraw the view windows.

Viewing the Deflected Shapes

Once the design has been performed, you may view the deflection shapes for any span at any available loading pattern.

To view the deflection shapes:

1. Select the Deflection command from the **View** menu or click the  button on the tool bar. A view window similar to Figure 4-69 will appear.

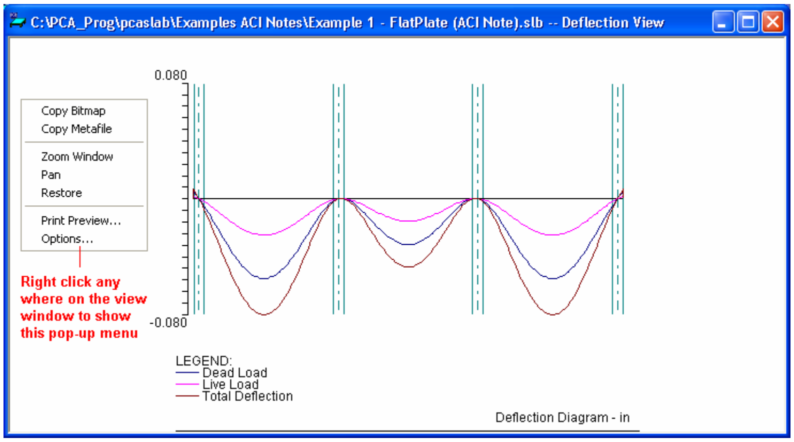


Figure 4-69 View Deflection Diagram

2. Right click on the **Deflection** view window and select **Options** command from the pop-up menu. A dialog box similar to Figure 4-70 will appear.

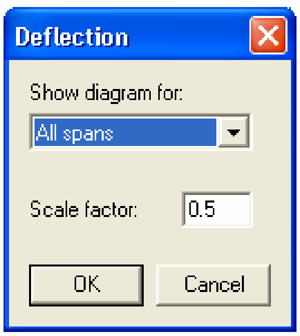


Figure 4-70 View Deflection Option dialog box

3. The current coordinate values can be captured based on the position of the mouse curser. The status bar shows the name of the diagram, two coordinate values of the current mouse curser position, and the current design code used in the project.
4. Click the right mouse button anywhere on the view window to show the pop-up menu. You may **Restore**, **Zoom**, **Pan** or **Print View** directly by selecting commands from this pop-up menu.
5. Click the **Options** command from the pop-up menu to change the span for which the deflection will be shown. Select the span you want to show from the **Show diagram for** drop-down list. Enter the scale factor in the **Scale factor** edit box. The bigger the scale factor, the more apparent the deflections will be on the diagram.
6. Press the **OK** button to close the dialog box and redraw the view windows.

Print Results

Printing the Analysis and Design Results

Once the analysis and/or design is performed, you can print the results. This section provides procedures for performing these functions.

To print the analysis and design results:

1. Select **Report** command from the **Solve** menu or click the  button from the tool bar. A dialog box similar to Figure 4-71 will appear.

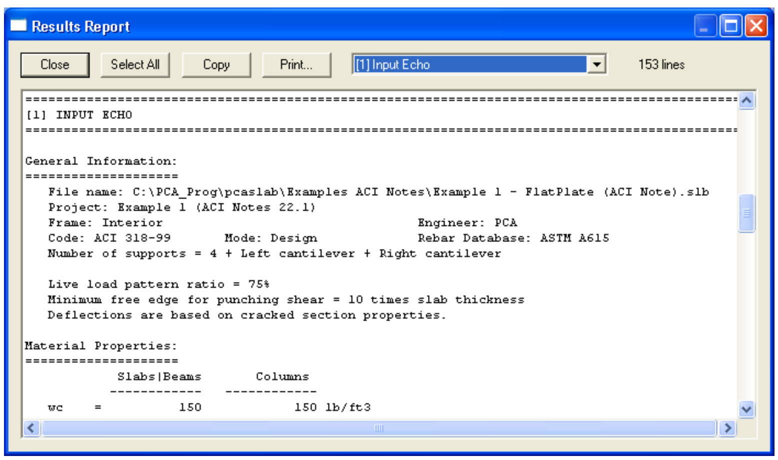


Figure 4-71 View and Print Results dialog box

2. Select the results you want to view from the drop-down list. The contents of the results list box will be changed based on your selection.
3. Press the **Print** button on the dialog box to print the results through a printer. The printer could be a local printer which is connected to your computer directly, or a network printer.
4. Press the **Close** button to close the result dialog box.

Printing the Current View Window

Once the design has been performed, you may print the diagrams and views for any span at any available loading pattern by selecting the **Print View** command from the **File** menu.

To print a displaying window:

1. To select the diagram or view you want to print, single click the left mouse button on the diagram or view window.
2. Select the **Print View** command from the **File** menu. A print preview window similar to Figure 4-72 will be shown.

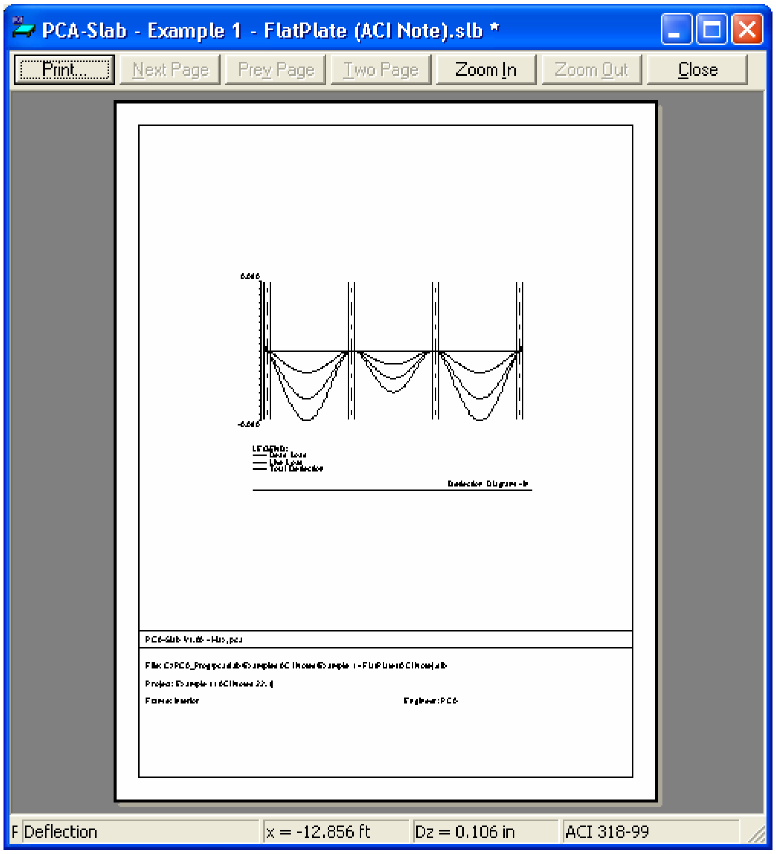


Figure 4-72 Print Preview of a View Window

3. Press the **Zoom In** or **Zoom Out** buttons or simply click the left mouse button on the preview to magnify or reduce the size of the preview paper.
4. Press the **Next Page** button if more than one page need to be printed.
5. Press the **Print** button to print the view. The printer could be a local printer which is connected to your computer directly, or a network printer.
6. Press the **Close** button to close the preview window and go back to **pcaSlab**.

Print Preview

Print Preview

The **Print Preview** command allows you to preview and print the current view window (floor system geometry in the plan, elevated and isometric views, prints the shear and moment diagrams, and the deflected shapes).

1. To obtain a view window you must first perform the design, then select what you want to view from the **View** menu. You may have more than one view window opened. The current view window is the one activated and on top of the others on your screen.
2. Selecting this command closes the **pcaSlab** main window and opens the print preview window as shown in Figure 4-73.
3. On the print preview window, press the **Zoom In** or **Zoom Out** buttons or simply click the left mouse button on the preview window to magnify or reduce the size of the preview paper.
4. Press the **Next Page** button if more than one page needs to be printed.
5. Press the **Print** button to print the view. The printer could be a local printer which is connected to your computer directly, or a network printer.
6. Press the **Close** button to close the preview window and go back to **pcaSlab**.

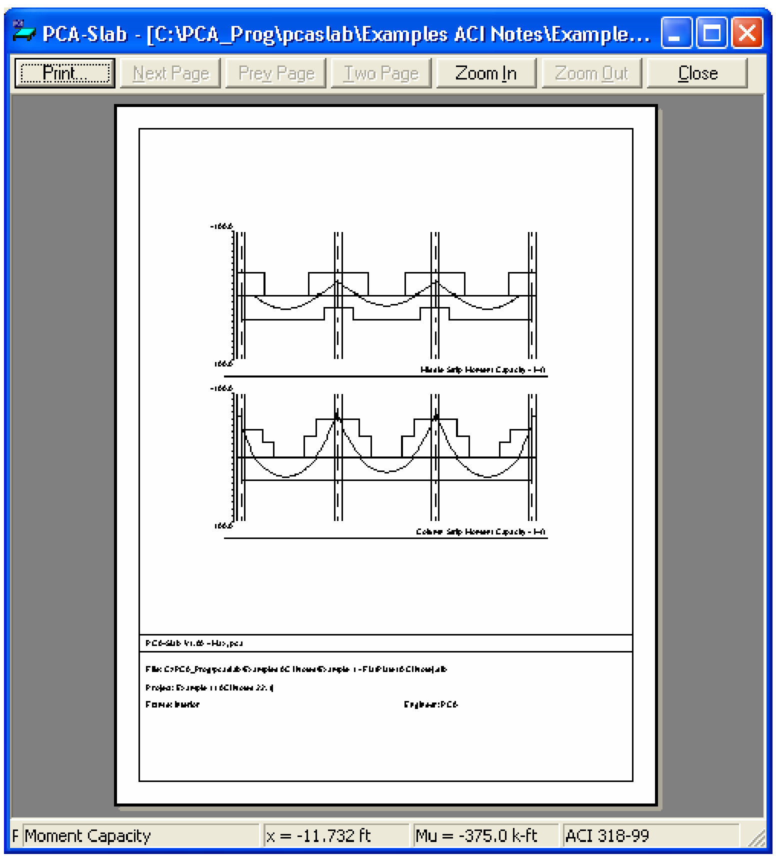


Figure 4-73 Print Preview Window

Copy Graphs to Clipboard

Copy Bitmap (BMP format)

One of the nice things about the Windows operating system is that it supplies a standard bitmapped image file format, the BMP. As its name implies, the Windows bitmap format is a direct bitmap representation of an image. **pcaSlab** can copy any of the ten view windows onto Windows clipboard as bitmap. The bitmap on clipboard can then be pasted into Microsoft Word file or other Windows word processing software, as well as presentation software such as Microsoft PowerPoint.

To copy view window to clipboard as bitmap:

1. Select the view window that will be copied by single clicking left mouse button on it.
2. Select  from the tool bar to copy the selected view window to clipboard.
3. Switch to other word processing software, such as Microsoft Word, then press the **Ctrl + V** to paste the bitmap on clipboard to a Word file.

Copy Metafile (EMF format)

Since bitmap files cannot easily be resized or re-proportioned without significant distortion to the image, metafiles are generally used for situations requiring scalability of the image.

Advantages of metafiles are:

1. Large, simply structured images require less memory than bitmaps for display and make optimal use of the resolution of the output device.
2. Metafiles can be resized with none of the distortion which normally accompanies resizing of bitmaps.
3. A metafile can contain SelectPalette statements, allowing custom palettes to be displayed in applications such as Microsoft Word. (According to Microsoft, this may result in lost resources in 16 bit Windows operating systems.).

The Enhanced MetaFile (EMF) format is an extension of the Windows metafiles format developed for use with 32 bit Windows applications. It is only available to native 32 bit applications.

To copy view window to clipboard as Enhanced MetaFile (EMF):

1. Select the view window that will be copied by single clicking left mouse button on it.
2. Select  from the tool bar to copy the selected view window to clipboard.
3. Switch to other word processing software, such as Microsoft Word, then press the **Ctrl + V** to paste the bitmap on clipboard to a Word file.

Customizing Program (menu Options)

Changing Colors

Colors can be changed for background of views, geometry items such as slabs and beams, text on views, result diagrams, etc.

To change colors:

1. Select **Colors** command from the **Options** menu. Figure 4-74 will appear.

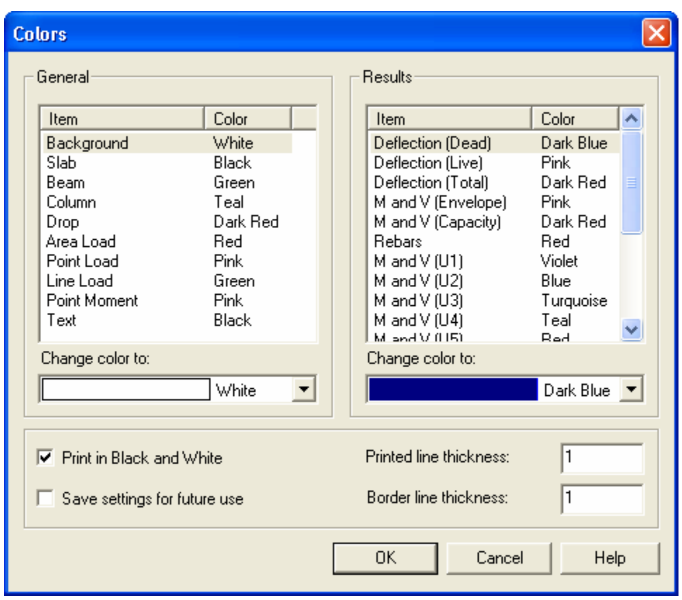


Figure 4-74 Changing Colors Dialog Box

2. From the **General** frame box on the left side, select the item whose color needs to be changed from the list box.
3. Select a color from the **Change color to** drop-down list. Once a new color is selected from the drop-down list the color in the list box above the drop-down list is updated instantly.
4. From the **Results** category box on the right side, select the item whose color needs to be changed from the list box.
5. Select a color from the **Change color to** drop-down list. Once a new color is selected from the drop-down list the color in the list box above the drop-down list is updated instantly.
6. If you want to print views in black and white, select the **Print Black and White** check box.
7. If you want to save the settings as default, select the **Save setting for future use** check box.
8. Input the line thickness in the **Printed Line Thickness** edit box. The diagram line thickness will be based on the number you input.
9. Press the **OK** button to save the settings and close the dialog box.

Changing Startup Defaults

To change the startup defaults:

1. Select the **Startup Defaults** command from the **Options** menu. A view window similar to Figure 4-75 will appear.

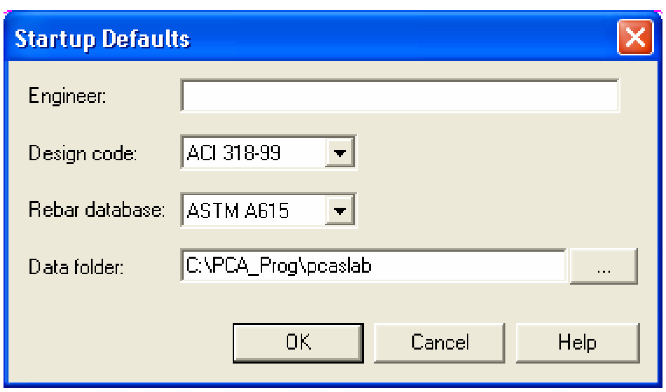


Figure 4-75 Changing Startup Defaults Dialog Box

2. Input the name of the engineer in the **Engineer** edit box. The name of the engineer will be shown on the view print.
3. Select the design code from the **Design code** drop-down list. Four design codes are available: ACI 318-99, ACI 318M-99, CSA A23.3-94 and CSA A23.3-94E.
4. Select the rebar database from the **Rebar database** drop-down list. Four defined

databases are available: ASTM A615, ASTM A615M, CSA G30.18 and prEN 10080. If you would like to use rebar defined by yourself, select the User Defined from the drop-down list. You may define your own rebar database by selecting the Rebar Database from the Options menu.

5. Press the **OK** button to save the new setting and close the dialog box.

Changing the Rebar Database

User can define his own rebar database. The defined database, such as ASTM A615M, cannot be changed.

To change the rebar database:

1. Select the **Rebar** database from the **Options** menu. Figure 4-76 will appear.

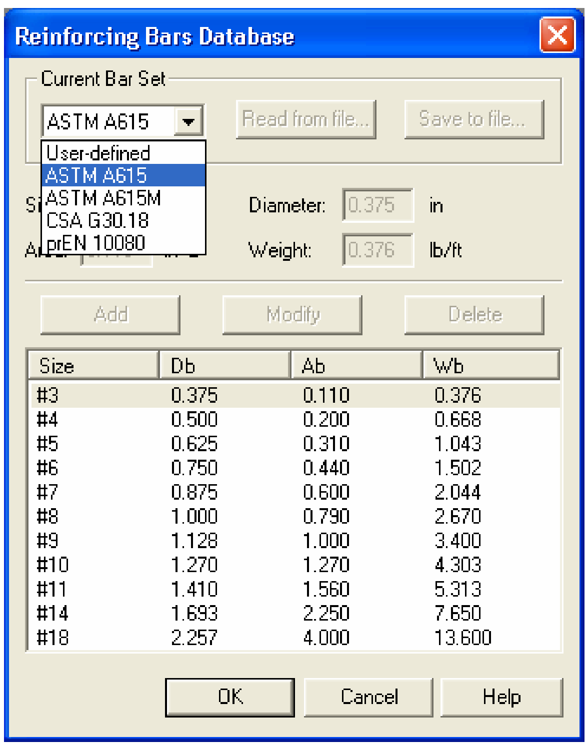


Figure 4-76 Changing Rebar Database

2. Select **User-defined** from the **Current Bar Set** drop-down list. Rebar data of the other databases can only be viewed and cannot be changed.
3. To select the rebar that needs to be changed from the rebar list box, single click the left mouse button on it. The size, diameter, area, and weight of the selected rebar are shown in the corresponding text boxes, respectively.
4. Enter the new values of bar size, diameter, area, and weight.
5. Press the **Add** button to add the bar as a new bar into the rebar list box on the

lower part of the dialog box. If any data inconsistency is found by **pcaSlab**, an error of "Inconsistent bar data" will be shown.

6. Press the **Modify** button to update an existing rebar.
7. To delete a rebar, select the rebar from the rebar list box then click the **Delete** button.
8. After finishing the definitions of your own rebar database, you may save them into a file on the hard drive by pressing the **Save to file** button and specifying a file name for your database. This file can be imported into **pcaSlab** by pressing the **Read from file** button.

Working with View Windows (menu Window)

Cascade

The **Cascade** command displays all the open windows in the same size, arranging them on top of each other so that the title bar of each is visible. The current active view widow will be on the top after the execution of the **Cascade** command.

Tile Horizontal

The **Tile Horizontal** command arranges all open windows horizontally so that no window overlaps another. The current active view widow will be on the most left or on the upper-left corner of the screen after the execution of the **Tile Horizontal** command.

Tile Vertical

The **Tile Vertical** command arranges all open windows vertically so that no window overlaps another. The current active view widow will be on the most left or on the upper-left corner of the screen after the execution of the **Tile Vertical** command.

Remaining Commands

The remaining menu items are in a list of the windows that are available for viewing. Selecting any window from this menu will restore the window to its previous size and position from an icon.

Obtaining Help Information (menu Help)

Opening Help in Contents mode

To open Help in Contents mode:

1. Select **Contents** from the **Help** menu.
2. Select the chapter you need from the content tree view in the left pane of the Help window. The contents of the selected chapter appear in the right pane of the Help window.

Opening Help in Index Mode

To open Help in Index mode:

1. Select **Index** from the **Help** menu.
2. Enter the key word you want to find into the "Type in the Keyword to Find" text box. The most close chapter will be highlighted in the left list box on the Help Window. Double left click the selected chapter to show the contents in the right pane.

Obtaining the Program Version

The **About** box contains the copyright information, the version number of the **pcaSlab** program, your firm name, city, and state.

To obtain the program version:

1. Select the **About pcaSlab** command from the **Help** menu. The dialog box of Figure 4-77 will appear.

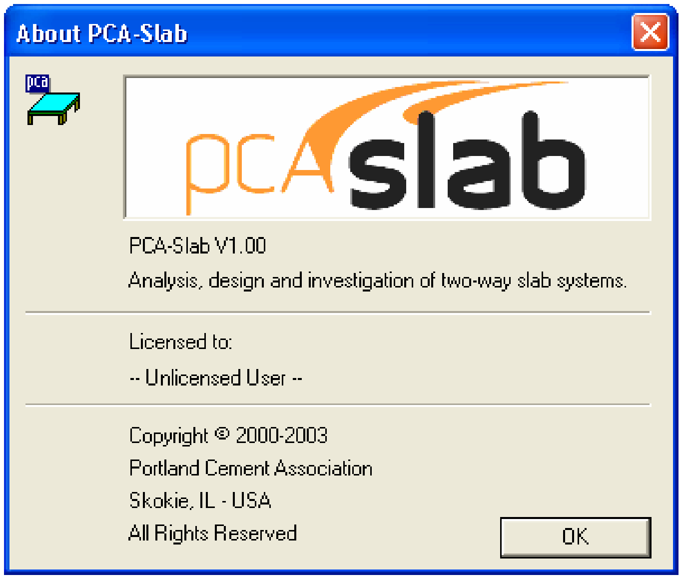


Figure 4-77 About pcaSlab dialog box

2. Press **Ok** button to exit the dialog box.

Ch. 5 Output Description

Output Elements

pcaSlab generates the text and graphical output of the input data and the results of the calculations. The text output is generated when user opens the **Results Report** dialog window. An ASCII text file is generated in the same sub-folder as the input data file. The name of the output file is created by adding the extension ".OUT" to the name of the input data file. Depending on the report options selected, the text output will contain a selection of the following sections (see the illustrated examples in the following chapter):

- Program Version
- [1] Input Echo
- [2] Design Results
- [3] Column Axial Forces And Moments
- [4] Segmental Moment And Shear - Load Cases
- [5] Segmental Moment And Shear - Load Combinations
- [6] Segmental Moment And Shear – Envelopes
- [7] Segmental Deflections

Program Version

The program version number appears at the top of each report page along with the copyright and disclaimer information.

Input Echo

Section **Input Echo** reports the data used in the analysis. **pcaSlab** defaults common data; all other data must be input. Carefully check the contents of the section and compare it with the intended design model. The following paragraphs describe the blocks included in the section.

General Information

This block is similar in its content to the dialog window **General Information**. It contains the information on project input data file name, project description, selected design code and units, selected rebar database, calculation mode (design or investigation), number of supports, cantilevers. The selections available in dialog window **Solve Options** are also listed in this block.

Material Properties

This block contains the information on concrete properties for slabs, beams and columns. It also contains the information on reinforcing steel properties for slabs and beams.

Rebar Database

This block lists the properties of the bars from the bar table selected for the project. Bar diameter, cross-section area and unit weight for each bar are reported. The values reported are consistent with the units used in particular model.

Span Data

This block is similar in its content to the dialog window **Span Data**. The block is divided into two parts. First part reports the span-by-span geometry of the concrete slab (length, left and right side width, depth and code required minimum thickness). The second part contains the span-by-span geometry of longitudinal beams and ribs (for waffle slabs).

Support Data

This block is similar in its content to the dialog window **Support Data**. The block is divided into four parts. The first part reports the geometry of top and bottom columns and the stiffness share factor. For circular column the transverse dimension C2 is reported as zero. The second part contains the geometry of drop panels: thickness, lengths, widths. If dimensions of a drop panel are invalid it will be marked. Invalid or excessive drop panel geometry is not used in the analysis. The third part contains the geometry of column capitals: depth, slope (depth/extension ratio), extensions. The fourth part contains the geometry of transverse beams: width depth, eccentricity (offset) from column centroid.

Load Data

This block contains the complete information on load input. The block is divided into three parts. The first part reports the defined load cases, load combinations and corresponding load factors. This part summarizes the contents of the dialog windows **Load Cases** and **Load Combinations**. The second part reports the magnitudes of defined span loads. It summarizes the contents of the dialog window **Span Loads**. The third part reports the magnitudes of lateral actions (joint moments) if defined in the model. It summarizes the contents of the dialog window **Lateral Load Effects**.

Reinforcement Criteria

This block is similar in its content to the dialog window **Reinforcement Criteria**. The block is divided into three parts. The first part reports the requirements for slab and rib bars. The second part reports the requirements for longitudinal beams. Both parts contain the information on bar sizes, covers, spacings and user selected allowable steel percentages. The requirements for top and bottom bars are given. For longitudinal beams additionally the criteria for transverse bars (stirrups) are listed.

Reinforcing Bars

This block is available only when Investigation Mode is selected. This block is similar in its content to the dialog window **Reinforcing Bars**. The block is divided into three parts. The first part reports the span-by span user selected top bars for column, middle and beam strips accordingly. Similarly, the second part reports the user selected bottom bars for column, middle and beam. For longitudinal bars the program reports bar sizes, lengths and concrete cover. The third part presents the beam transverse reinforcement (stirrups) defined by the user.

Note:

When switching from Design Mode to Investigation Mode, **pcaSlab** automatically assumes the results of the Design Mode as an input for Investigation Mode.

Design Results

Section **Design Results** presents the summary of the design results of the slab system. The following paragraphs describe the blocks included in the section.

Top Reinforcement

This block is available only when Design Mode is selected. It reports the negative reinforcement requirements. The block contains the values of corresponding design strip widths (column, middle, and beam), maximum factored design moments per strip and critical location, minimum and maximum steel areas, bar spacings, steel areas required by ultimate condition, selected bar sizes and numbers. The quantities are given for left, center and right location of each span.   For a detailed discussion, see Chapter 2, "Area of Reinforcement".

Note: This block does not include reinforcement quantities necessary to transfer negative unbalanced moment at supports.

Top Bar Details

The block contains a span-by-span listing of the longitudinal bars selected in column, middle and beam strips. This reinforcement schedule is intended as a guide for bar placement. In more complex cases the bar schedule selected by the program may have to be adjusted by the user for constructability reasons. The selected bar sizes are limited by user specified minimum and maximum sizes. Bar sizes and numbers are selected to satisfy the minimum and required steel areas in conjunction with the bar spacing requirements of the Code. The program calculates the bar lengths based on the computed inflection points and the recommended minimums of the Code. The bar lengths are adjusted by appropriate development lengths. Hooks and bends are not included in bar length tables and figures. For beams bars are placed in single a layer (see Figure 2-21), provided there is sufficient beam width. For a detailed discussion, see Chapter 4, "Reinforcement Selection".

Note: This block does not include reinforcement bars necessary to transfer negative unbalanced moment at supports.

Bottom Reinforcement

This block is available only when Design Mode is selected. It reports the positive reinforcement requirements. The block contains the values of corresponding design strip widths (column, middle, and beam), maximum factored design moments per strip and critical location, minimum and maximum steel areas, bar spacings, steel areas required by ultimate condition, selected bar sizes and numbers. The quantities are given for mid-span regions of each span. For a detailed discussion, see Chapter 2, "Area of Reinforcement".

Bottom Bar Details

This block contains a span-by-span listing of the longitudinal bars selected in column, middle and beam strips. The reinforcement schedule is intended as a guide for bar placement. In more complex cases the bar schedule selected by the program may have to be adjusted by the user for constructability reasons. The selected bar sizes are limited by user specified minimum and maximum sizes. Bar sizes and numbers are selected to satisfy the minimum and required steel areas in conjunction with the bar spacing requirements of the Code. The program calculates the bar lengths based on the computed inflection points and the recommended minimums of the Code. The bar lengths are adjusted by appropriate development lengths. Hooks and bends are not included in bar length tables and figures. For beams bars are placed in single a layer (see Figure 2-21), provided there is sufficient beam width. For a detailed discussion, see Chapter 2, "Reinforcement Selection".

Flexural Capacity

This block lists the selected top and bottom steel areas and corresponding negative and positive moment capacity values in each span. The data is subdivided between column, middle and beam strips. Each span is subdivided into segments reflecting the changes in geometry and bar placement.

Longitudinal Beam Shear Reinforcement Required

This block is available only when Design Mode is selected. It reports the requirements of transverse reinforcement for each longitudinal beam. The capacity of concrete cross-section ϕV_c in each span is shown. The table contains the segmental values of the factored shear force Vu and required intensity of stirrups (A_v/s). The segmental values cover the distance between left and right critical sections, and include locations where there is change of geometry of loading.

Longitudinal Beam Shear Reinforcement Details

This block is available only when Design Mode is selected. It is intended as a guide for stirrup placement. The output presents the program selected stirrup sizes, numbers and spacings. Distances between groups of stirrups are also reported.

Beam Shear Capacity

This block lists the concrete section shear capacity ϕVc, selected stirrup intensities and spacings and corresponding beam shear capacity ϕVn values in each span. The maximum factored shear forces V_{max} in beam strip along the span also reported.

Slab Shear Capacity

This block lists the values of one-way slab shear capacity ϕVc in each span. The maximum factored shear force Vu and the location of the critical section X_u are also reported.

Flexural Transfer of Negative Unbalanced Moment at Supports

This block reports the design values for negative reinforcement necessary to transfer unbalanced support moments. The block contains the results for critical (effective) section width as per the Code, the maximum unbalanced moment, the corresponding load combination and governing load pattern, the reinforcing steel areas provided and additional steel required. The provided reinforcement area (main longitudinal bars) is reduced by the ratio of critical (effective) strip width to total strip width and does not include the required area due to unbalanced moments. The additional reinforcement is the difference between that required by unbalanced moment transfer by flexure and that provided for design bending moment. When additional reinforcement is required, it is selected based on the bar sizes already provided at the support. For a detailed discussion, see Chapter 2, "Area of Reinforcement" and "Additional Reinforcement at Support".

Punching Shear Around Columns

The block contains the values pertaining to punching shear check in critical sections around the columns. The table lists two sets of punching shear calculations – direct shear alone and direct shear with moment transfer. The output contains the values of the allowable shear stress ϕv_c, reactions Vu, unbalanced moments M_{unb}, governing load pattern, fraction of unbalanced moment □v, punching shear stress vu. The calculation for moment transfer adjusts the unbalanced moment to the centroid of the critical section. The "shear transfer" is the unbalanced moment multiplied by ϕv. When calculated shear stress vu exceeds the allowable value ϕv_c, the program prints a warning flags for this support. For a detailed discussion, see Chapter 2, "Shear Analysis of Slabs".

Punching Shear Around Drops

The block contains the values pertaining to punching shear check in a critical section around the drop panels. The table displays the reactions V_u, governing load pattern, the punching shear stress around the drop v_u, and the allowable shear stress ϕvc. When calculated shear stress v_u exceeds the allowable value ϕvc, the program prints a warning flags for this drop panel. For a detailed discussion, see Chapter 2, "Shear Analysis of Slabs".

Maximum Deflections

This block lists the summary of dead load (DL), live load (LL) and total (DL+LL) short-time deflections for the entire equivalent frame, column and middle strips. If solution option "Gross (uncracked) sections" is selected, the values of deflections reported are based on gross section properties. If solution option "Effective (cracked) sections" is used, the values of deflections reported are based on substitute effective moment of inertia of the section. For a detailed discussion, see Chapter 2, "Deflection Calculation".

Material Takeoff

This block lists the approximate total and unit quantities of concrete, and reinforcement. Note that the reinforcement estimate is for one direction only and ignores items such as hooks, bends, and waste. For a detailed discussion, see Chapter 2, "Material Quantities".

Column Axial Forces And Moments

Section **Column Axial Forces And Moments** presents the summary of unfactored axial forces (reactions) and bending moments in bottom and top columns in the column-slab joints. The values reported represent the loading of a single floor only. Any actions on the columns from the floors above must be added to this story's actions to properly analyze/design the columns. The output contains column actions due to Selfweight, Dead Load, Live Load, and Total Combination. The Live Load values for all four load patterns are presented. The values for Live Load pattern No.4 are not combined because this load pattern is not singular (each support has is individual load configuration).

Segmental Moment And Shear - Load Cases

This section presents the summary of unfactored bending moments and shear forces for individual load cases including selfweight, dead load, live load and lateral cases. The reported values are presented using span-by-span segmental approach.

Segmental Moment And Shear - Load Combinations

This section presents the summary of bending moments and shear forces for each load combination. The reported values for each load combination are presented using span-by-span segmental approach. The negative and positive values of bending moments and shear forces are presented in separate columns in order to provide consistent format with enveloped output.

Segmental Moment And Shear - Envelopes

This section presents the summary of bending moments and shear forces for envelope of all load combinations. The reported values are presented using span-by-span segmental approach. The negative and positive values of bending moments and shear forces are presented in separate columns for user convenience. The factored values presented in this section are used for design purposes (longitudinal and transverse reinforcement).

Segmental Deflections

This section presents the summary of deflections for unfactored (service) load cases including selfweight and dead load (DL), live load (LL) and combined (DL+LL) load cases. The reported values are presented using span-by-span segmental approach. If solution option "Gross (uncracked) sections" is selected, the values of deflections reported are based on gross section properties. If solution option "Effective (cracked) sections" is used, the values of deflections reported are based on substitute effective moment of inertia of the section. For a detailed discussion, see Chapter 2, "Deflection Calculation".

Graphical Output

pcaSlab provides the following graphical output features:

- Shear and Moment Diagrams
- Moment Capacity Diagram
- Shear Capacity Diagram
- Deflection Diagram
- Reinforcement Diagram

These diagram windows can be customized. The options dialog allows selecting either a single span or all spans. Other elements of the graphs can also be modified. **pcaSlab** print preview of the current graphical window. The user has also the choice to export the graphics to a metafile or bitmap file.

Detailed information on using the graphical output features is included in chapter "Operating the Program".

Ch. 6 Examples

Flat Slab

Problem Description

Analyze and design an interior strip shown in Figure 6-1.

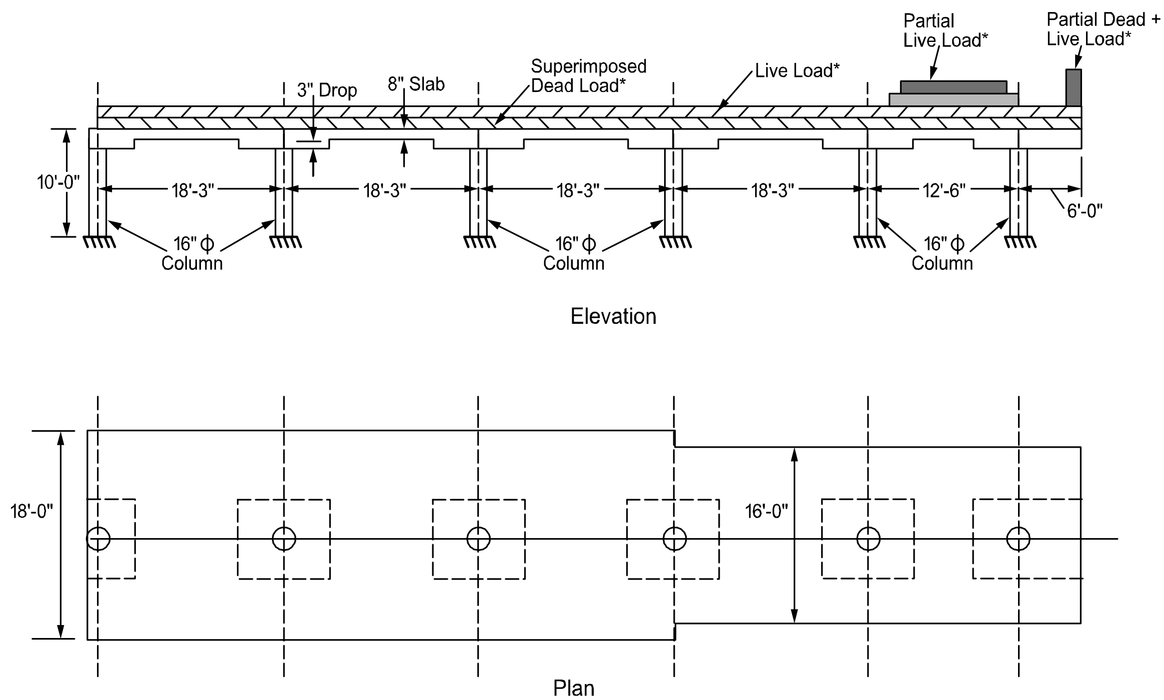


Figure 6-1 Example flat slab problem

Calculation Results - Text Output

The following listing contains the first three sections of the result report for Example 1.

```
                 ooooooo      oooooo      ooooo
                 oooooooo    oooooooo    ooooooo
                 oo     oo   oo     oo   oo    oo
                 oo     oo   oo          oo    oo
                 oooooooo    oo          ooooooo    ooooo
                 ooooooo     oo     oo   ooooooo    ooooo
                 oo          oo     oo   oo    oo
                 oo          oooooooo    oo    oo
                 oo           oooooo     oo    oo


               oooooo      o                    o
              oooooooo    oo        ooooo      oo
              oo          oo       o     oo    oo
              oooo        oo       o     oo    oo
               oooooo     oo        oooooo     oooooo
                  oooo    oo       oo    oo    oo    oo
                    oo    oo       oo    oo    oo    oo
              oooooooo    oo   o   oo    oo    oo    oo
               oooooo      ooo      ooooo o     ooooo
        ============================================================================
                            pcaSlab V1.01 (TM)
                A Computer Program for Analysis and Design of Slab Systems
        ============================================================================
                       Copyright © 2000-2003, Portland Cement Association
                                       All rights reserved

              Licensee stated above acknowledges that Portland Cement Association
           (PCA) is not and  cannot  be  responsible for  either the  accuracy  or
           adequacy of  the material supplied  as input  for processing   by  the
           pcaSlab computer program.  Furthermore, PCA neither makes any warranty
           expressed nor implied with respect  to  the  correctness of the  output
           prepared by the  pcaSlab  program.  Although  PCA  has  endeavored  to
           produce pcaSlab error free the  program is not and cannot be certified
           infallible.  The final and only responsibility for analysis, design and
           engineering documents is the licensees. Accordingly, PCA disclaims  all
           responsibility in contract, negligence or other tort for any  analysis,
           design  or engineering documents prepared in connection with the use of
           the pcaSlab program.


==============================================================================================
[1] INPUT ECHO
==============================================================================================

General Information:
====================
   File name: C:\Program Files\PCA\pcaslab\Examples Adoss\fltslb.slb
   Project: FLAT SLAB -INTERIOR BENT-VERTICAL
   Frame: Adoss Example 2                          Engineer: PCA
   Code: ACI 318-99       Mode: Design             Rebar Database: ASTM A615
   Number of supports = 6 + Left cantilever + Right cantilever

   Live load pattern ratio = 100%
   Minimum free edge for punching shear = 10 times slab thickness
   Deflections are based on cracked section properties.

Material Properties:
====================
            Slabs|Beams       Columns
           ------------   ------------
   wc    =          150           150 lb/ft3
   f'c   =            4             4 ksi
   Ec    =      3834.25       3834.25 ksi
   fr    =       0.4743        0.4743 ksi

   fy    =           60 ksi, Bars are not epoxy-coated
   fyv   =           60 ksi
   Es    =        29000 ksi

Rebar Database:
===============
   Units: Db (in), Ab (in^2), Wb (lb/ft)
   Size       Db       Ab       Wb     Size       Db       Ab       Wb
   ---- -------- -------- --------     ---- -------- -------- --------
     #3     0.38     0.11     0.38       #4     0.50     0.20     0.67
     #5     0.63     0.31     1.04       #6     0.75     0.44     1.50
     #7     0.88     0.60     2.04       #8     1.00     0.79     2.67
     #9     1.13     1.00     3.40      #10     1.27     1.27     4.30
    #11     1.41     1.56     5.31      #14     1.69     2.25     7.65
```

```
    #18     2.26     4.00    13.60

Span Data:
==========
   Slabs: L1, wL, wR (ft); t (in)
   Span Loc        L1        t       wL       wR     Hmin
   ---- ---- -------- -------- -------- -------- --------
      1 Int     0.670     8.00    9.000    9.000     7.20 LC
      2 Int    18.300     8.00    9.000    9.000     6.22
      3 Int    18.300     8.00    9.000    9.000     5.71
      4 Int    18.300     8.00    9.000    9.000     6.22
      5 Int    18.300     8.00    8.000    8.000     6.22
      6 Int    12.500     8.00    8.000    8.000     5.39
      7 Int     6.000     8.00    8.000    8.000     6.40 RC

Support Data:
=============
   Columns: c1a, c2a, c1b, c2b (in); Ha, Hb (ft)
   Supp      c1a      c2a       Ha        c1b      c2b       Hb  Red%
   ---- -------- -------- --------   -------- -------- --------  ----
      1     0.00     0.00    0.000      16.00     0.00   10.000   100
      2     0.00     0.00    0.000      16.00     0.00   10.000   100
      3     0.00     0.00    0.000      16.00     0.00   10.000   100
      4     0.00     0.00    0.000      16.00     0.00   10.000   100
      5     0.00     0.00    0.000      16.00     0.00   10.000   100
      6     0.00     0.00    0.000      16.00     0.00   10.000   100

   Drop Panels: h (in); L1, L2, W1, W2 (ft)
   Supp        h       L1       L2       W1       W2
   ---- -------- -------- -------- -------- --------
      1     3.00    0.670    3.050    3.000    3.000 *d
      2     3.00    3.050    3.050    3.000    3.000
      3     3.00    3.050    3.050    3.000    3.000
      4     3.00    3.050    3.050    2.667    2.667 *c d
      5     3.00    3.050    2.083    2.667    2.667
      6     3.00    2.083    6.000    2.667    2.667
   *c- Invalid drop. Drop thickness will not be used for flexural design.
   *d- Excessive drop thickness will not be used for flexural design.

Load Data:
==========
   Load Cases and Combinations:
   Case     SELF     Dead     Live
   Type     DEAD     DEAD     LIVE
   U1      1.400    1.400    1.700

   Span Loads:
   Span Case               Wa           La           Wb           Lb
   ---- -------- ------------ ------------ ------------ ------------
   Line Loads - Wa | Wb (lb/ft), La | Lb (ft):
      7 Dead              500          5.5          500            6
      6 Live              800          2.5          800           10
      7 Live               50          5.5           50            6
   Area Loads - Wa (lb/ft2):
      2 Dead               10
      3 Dead               10
      4 Dead               10
      5 Dead               10
      6 Dead               10
      7 Dead               10
      2 Live               50
      3 Live               50
      4 Live               50
      5 Live               50
      6 Live              100
      7 Live               40

Reinforcement Criteria:
=======================
                 _____Top bars___  ___Bottom bars__  ____Stirrups____
                   Min      Max      Min      Max      Min      Max
                 -------  -------  -------  -------  -------  -------
   Slabs and Ribs:
     Bar Size         #4       #8       #4       #8
     Bar spacing    6.00    18.00     6.00    18.00 in
     Reinf ratio    0.18     2.00     0.18     2.00 %
     Cover          1.50              1.00          in

   Beams:
     Bar Size         #5       #8       #5       #8       #3       #5
     Bar spacing    2.00    18.00     2.00    18.00     6.00    18.00 in
     Reinf ratio    0.20     2.00     0.20     2.00 %
     Cover          1.50              1.50          in


==============================================================================================
[2] DESIGN RESULTS
==============================================================================================

Top Reinforcement:
==================
   Units: Width (ft), Mmax (k-ft), Xmax (ft), As (in^2), Sp (in)
```

Span	Strip	Zone	Width	Mmax	Xmax	AsMin	AsMax	SpReq	AsReq	Bars
1	Column	Left	9.00	0.00	0.001	1.555	13.500	13.500	0.000	8-#4
		Middle	9.00	0.00	0.002	1.555	14.013	0.000	0.000	8-#4
		Right	9.00	0.00	0.003	1.586	9.342	10.800	0.000	10-#4
	Middle	Left	9.00	0.00	0.001	1.555	13.500	13.500	0.000	8-#4
		Middle	9.00	0.00	0.002	1.555	13.500	13.500	0.000	8-#4
		Right	9.00	0.00	0.003	1.555	13.500	13.500	0.000	8-#4
2	Column	Left	9.00	29.69	0.667	1.944	13.320	10.800	0.720	10-#4
		Middle	9.00	0.00	0.000	0.000	0.000	0.000	0.000	---
		Right	9.00	103.72	17.633	1.944	13.320	8.308	2.580	13-#4
	Middle	Left	9.00	-0.00	0.667	1.555	13.500	13.500	0.000	8-#4
		Middle	9.00	0.00	0.000	0.000	0.000	0.000	0.000	---
		Right	9.00	34.58	17.633	1.555	13.500	13.500	1.250	8-#4
3	Column	Left	9.00	99.00	0.667	1.944	13.320	8.308	2.458	13-#4
		Middle	9.00	0.43	6.605	1.555	13.500	0.000	0.015	8-#4
		Right	9.00	84.43	17.633	1.944	13.320	9.818	2.086	11-#4
	Middle	Left	9.00	33.00	0.667	1.555	13.500	13.500	1.192	8-#4
		Middle	9.00	0.14	6.605	1.555	13.500	0.000	0.005	8-#4
		Right	9.00	28.14	17.633	1.555	13.500	13.500	1.014	8-#4
4	Column	Left	9.00	85.40	0.667	1.944	13.320	9.818	2.111	11-#4
		Middle	9.00	0.00	0.000	0.000	0.000	0.000	0.000	---
		Right	8.00	85.58	17.633	1.382	11.880	8.727	3.228	11-#5
	Middle	Left	9.00	28.47	0.667	1.555	13.500	13.500	1.026	8-#4
		Middle	9.00	0.00	0.000	0.000	0.000	0.000	0.000	---
		Right	10.00	28.53	17.633	1.728	15.000	13.333	1.027	9-#4
5	Column	Left	8.00	85.74	0.667	1.382	11.880	8.727	3.235	11-#5
		Middle	8.00	0.00	0.000	0.000	0.000	0.000	0.000	---
		Right	6.25	71.16	17.633	1.426	11.840	8.333	1.755	9-#4
	Middle	Left	8.00	28.58	0.667	1.382	12.000	13.333	1.032	9-#4
		Middle	8.00	0.00	0.000	0.000	0.000	0.000	0.000	---
		Right	9.75	23.72	17.633	1.685	14.625	13.000	0.852	9-#4
6	Column	Left	6.25	65.37	0.667	1.426	11.840	8.333	1.609	9-#4
		Middle	6.25	14.23	4.575	1.080	9.375	0.000	0.511	6-#4
		Right	6.25	54.69	11.833	1.426	11.840	9.375	1.341	8-#4
	Middle	Left	9.75	21.79	0.667	1.685	14.625	13.000	0.782	9-#4
		Middle	9.75	2.76	4.575	1.685	14.625	0.000	0.098	9-#4
		Right	9.75	-0.00	11.833	1.685	14.625	13.000	0.000	9-#4
7	Column	Left	6.25	56.52	0.667	1.426	11.840	9.375	1.386	8-#4
		Middle	6.25	24.30	2.533	1.080	13.875	9.375	0.588	8-#4
		Right	6.25	7.34	4.133	1.080	9.375	9.375	0.262	8-#4
	Middle	Left	9.75	0.00	0.667	1.685	14.625	13.000	0.000	9-#4
		Middle	9.75	0.00	2.533	1.685	14.625	13.000	0.000	9-#4
		Right	9.75	0.00	4.133	1.685	14.625	13.000	0.000	9-#4

Top Bar Details:
================

Units: Length (ft)

		Left				Continuous		Right			
Span	Strip	Bars	Length	Bars	Length	Bars	Length	Bars	Length	Bars	Length
1	Column	---		---		8-#4	0.67	2-#4	0.67	---	
	Middle	---		---		8-#4	0.67	---		---	
2	Column	5-#4	6.27	5-#4	4.06	---		7-#4	6.60	6-#4	4.06
	Middle	8-#4	4.40	---		---		8-#4	6.60	---	
3	Column	5-#4	6.27	---		8-#4	18.30	3-#4	6.27	---	
	Middle	---		---		8-#4	18.30	---		---	
4	Column	6-#4	6.60	5-#4	4.06	---		6-#5	6.60	5-#5	4.06
	Middle	8-#4	6.60	---		---		9-#4	6.60	---	
5	Column	6-#5	7.10	5-#5	4.06	---		5-#4	6.27	4-#4	4.06
	Middle	9-#4	7.10	---		---		9-#4	6.10	---	
6	Column	3-#4	4.35	---		6-#4	12.50	2-#4	4.35	---	
	Middle	---		---		9-#4	12.50	---		---	
7	Column	---		---		8-#4	6.00	---		---	
	Middle	---		---		9-#4	6.00	---		---	

Bottom Reinforcement:
====================

Units: Width (ft), Mmax (k-ft), Xmax (ft), As (in^2), Sp (in)

Span	Strip	Width	Mmax	Xmax	AsMin	AsMax	SpReq	AsReq	Bars
1	Column	9.00	0.00	0.000	0.000	14.580	0.000	0.000	---
	Middle	9.00	0.00	0.000	0.000	14.580	0.000	0.000	---

2	Column	9.00	52.46	8.030	1.555	14.580	12.000	1.765	9-#4
	Middle	9.00	34.97	8.030	1.555	14.580	13.500	1.168	8-#4
3	Column	9.00	35.21	9.523	1.555	14.580	13.500	1.176	8-#4
	Middle	9.00	23.47	9.523	1.555	14.580	13.500	0.780	8-#4
4	Column	9.00	39.78	9.026	1.555	14.580	13.500	1.331	8-#4
	Middle	9.00	26.52	9.026	1.555	14.580	13.500	0.882	8-#4
5	Column	8.00	35.46	9.523	1.382	12.960	13.714	1.187	7-#4
	Middle	8.00	23.64	9.523	1.382	12.960	13.714	0.787	7-#4
6	Column	6.25	29.22	6.495	1.080	10.125	12.500	0.979	6-#4
	Middle	9.75	19.48	6.495	1.685	15.795	13.000	0.646	9-#4
7	Column	6.25	0.00	0.000	0.000	10.125	0.000	0.000	---
	Middle	9.75	0.00	0.000	0.000	15.795	0.000	0.000	---

Bottom Bar Details:
===================

Units: Start (ft), Length (ft)

Span	Strip	Long Bars			Short Bars		
		Bars	Start	Length	Bars	Start	Length
1	Column	---			---		
	Middle	---			---		
2	Column	9-#4	0.00	18.30	---		
	Middle	4-#4	0.00	18.30	4-#4	0.00	15.56
3	Column	8-#4	0.00	18.30	---		
	Middle	4-#4	0.00	18.30	4-#4	2.75	12.81
4	Column	8-#4	0.00	18.30	---		
	Middle	4-#4	0.00	18.30	4-#4	2.75	12.81
5	Column	7-#4	0.00	18.30	---		
	Middle	4-#4	0.00	18.30	3-#4	2.75	12.81
6	Column	6-#4	0.00	12.50	---		
	Middle	5-#4	0.00	12.50	4-#4	1.88	10.63
7	Column	---			---		
	Middle	---			---		

Flexural Capacity:
==================

Units: From, To (ft), As (in^2), PhiMn (k-ft)

Span	Strip	From	To	AsTop	AsBot	PhiMn-	PhiMn+
1	Column	0.000	0.001	2.00	0.00	-54.78	0.00
		0.001	0.002	2.00	0.00	-56.92	0.00
		0.002	0.003	2.00	0.00	-56.18	0.00
		0.003	0.670	2.00	0.00	-56.18	0.00
	Middle	0.000	0.001	1.60	0.00	-44.06	0.00
		0.001	0.002	1.60	0.00	-44.06	0.00
		0.002	0.003	1.60	0.00	-44.06	0.00
		0.003	0.670	1.60	0.00	-44.06	0.00
2	Column	0.000	0.667	2.00	1.80	-81.04	53.48
		0.667	3.050	2.00	1.80	-81.04	53.48
		3.050	4.061	2.00	1.80	-54.78	53.48
		4.061	6.266	1.00	1.80	-27.76	53.48
		6.266	6.605	0.00	1.80	0.00	53.48
		6.605	11.695	0.00	1.80	0.00	53.48
		11.695	11.700	0.00	1.80	0.00	53.48
		11.700	14.239	1.40	1.80	-38.65	53.48
		14.239	15.250	2.60	1.80	-70.64	53.48
		15.250	17.633	2.60	1.80	-104.50	53.48
		17.633	18.300	2.60	1.80	-104.50	53.48
	Middle	0.000	0.667	1.60	1.60	-44.06	47.66
		0.667	4.400	1.60	1.60	-44.06	47.66
		4.400	6.605	0.00	1.60	0.00	47.66
		6.605	11.695	0.00	1.60	0.00	47.66
		11.695	11.700	0.00	1.60	0.00	47.66
		11.700	15.555	1.60	1.60	-44.06	47.66
		15.555	17.633	1.60	0.80	-44.06	24.06
		17.633	18.300	1.60	0.80	-44.06	24.06
3	Column	0.000	0.667	2.60	1.60	-104.50	47.66
		0.667	3.050	2.60	1.60	-104.50	47.66
		3.050	6.266	2.60	1.60	-70.64	47.66
		6.266	6.605	1.60	1.60	-44.06	47.66
		6.605	11.695	1.60	1.60	-44.06	47.66
		11.695	12.034	1.60	1.60	-44.06	47.66
		12.034	15.250	2.20	1.60	-60.10	47.66
		15.250	17.633	2.20	1.60	-88.91	47.66
		17.633	18.300	2.20	1.60	-88.91	47.66
	Middle	0.000	0.667	1.60	0.80	-44.06	24.06
		0.667	2.745	1.60	0.80	-44.06	24.06
		2.745	6.605	1.60	1.60	-44.06	47.66

		6.605	11.695	1.60	1.60	-44.06	47.66
		11.695	15.555	1.60	1.60	-44.06	47.66
		15.555	17.633	1.60	0.80	-44.06	24.06
		17.633	18.300	1.60	0.80	-44.06	24.06
4	Column	0.000	0.667	2.20	1.60	-88.91	47.66
		0.667	3.050	2.20	1.60	-88.91	47.66
		3.050	4.061	2.20	1.60	-60.10	47.66
		4.061	6.600	1.20	1.60	-33.22	47.66
		6.600	6.605	0.00	1.60	0.00	47.66
		6.605	11.695	0.00	1.60	0.00	47.66
		11.695	11.700	0.00	1.60	0.00	47.66
		11.700	14.239	1.86	1.60	-50.36	47.66
		14.239	17.633	3.41	1.60	-90.14	47.66
		17.633	18.300	3.41	1.60	-90.14	47.66
	Middle	0.000	0.667	1.60	0.80	-44.06	24.06
		0.667	2.745	1.60	0.80	-44.06	24.06
		2.745	6.600	1.60	1.60	-44.06	47.66
		6.600	6.605	0.00	1.60	0.00	47.66
		6.605	11.695	0.00	1.60	0.00	47.66
		11.695	11.700	0.00	1.60	0.00	47.66
		11.700	15.555	1.80	1.60	-49.55	47.66
		15.555	17.633	1.80	0.80	-49.55	24.06
		17.633	18.300	1.80	0.80	-49.55	24.06
5	Column	0.000	0.667	3.41	1.40	-90.14	41.71
		0.667	4.061	3.41	1.40	-90.14	41.71
		4.061	6.605	1.86	1.40	-50.36	41.71
		6.605	7.098	1.86	1.40	-50.36	41.71
		7.098	11.695	0.00	1.40	0.00	41.71
		11.695	12.034	0.00	1.40	0.00	41.71
		12.034	14.239	1.00	1.40	-27.60	41.71
		14.239	15.250	1.80	1.40	-48.91	41.71
		15.250	17.633	1.80	1.40	-72.91	41.71
		17.633	18.300	1.80	1.40	-72.91	41.71
	Middle	0.000	0.667	1.80	0.80	-49.28	24.04
		0.667	2.745	1.80	0.80	-49.28	24.04
		2.745	6.605	1.80	1.40	-49.28	41.71
		6.605	7.098	1.80	1.40	-49.28	41.71
		7.098	11.695	0.00	1.40	0.00	41.71
		11.695	12.198	0.00	1.40	0.00	41.71
		12.198	15.555	1.80	1.40	-49.53	41.71
		15.555	17.633	1.80	0.80	-49.53	24.04
		17.633	18.300	1.80	0.80	-49.53	24.04
6	Column	0.000	0.667	1.80	1.20	-72.91	35.69
		0.667	2.083	1.80	1.20	-72.91	35.69
		2.083	4.352	1.80	1.20	-48.91	35.69
		4.352	4.575	1.20	1.20	-32.99	35.69
		4.575	7.925	1.20	1.20	-32.99	35.69
		7.925	8.148	1.20	1.20	-32.99	35.69
		8.148	10.417	1.60	1.20	-43.64	35.69
		10.417	11.833	1.60	1.20	-65.01	35.69
		11.833	12.500	1.60	1.20	-65.01	35.69
	Middle	0.000	0.667	1.80	1.00	-49.53	30.04
		0.667	1.875	1.80	1.00	-49.53	30.04
		1.875	4.575	1.80	1.80	-49.53	53.58
		4.575	7.925	1.80	1.80	-49.53	53.58
		7.925	11.833	1.80	1.80	-49.53	53.58
		11.833	12.500	1.80	1.80	-49.53	53.58
7	Column	0.000	0.667	1.60	0.00	-65.01	0.00
		0.667	2.533	1.60	0.00	-65.01	0.00
		2.533	4.133	1.60	0.00	-65.24	0.00
		4.133	6.000	1.60	0.00	-43.64	0.00
	Middle	0.000	0.667	1.80	0.00	-49.53	0.00
		0.667	2.533	1.80	0.00	-49.53	0.00
		2.533	4.133	1.80	0.00	-49.53	0.00
		4.133	6.000	1.80	0.00	-49.53	0.00

Slab Shear Capacity:
====================

Units: b, d (in), Xu (ft), PhiVc, Vu(kip)

Span	b	d	Vratio	PhiVc	Vu	Xu
1	216.00	6.25	1.000	145.15	0.00	0.00
2	216.00	6.25	1.000	145.15	41.89	17.19
3	216.00	6.25	1.000	145.15	38.13	1.11
4	216.00	6.19	1.000	143.70	36.84	17.19
5	192.00	6.19	1.000	127.73	34.37	1.11
6	192.00	6.25	1.000	129.02	36.34	1.11
7	192.00	6.25	1.000	129.02	19.12	1.11

Flexural Transfer of Negative Unbalanced Moment at Supports:
==

Units: Width (in), Munb (k-ft), As (in^2)

Supp	Width	Munb	Comb	Pat	AsReq	AsProv	Additional Bars
1	49.00	32.30	U1	Even	1.143	0.907	2-#4
2	49.00	15.05	U1	Even	0.364	1.180	---
3	49.00	11.25	U1	Even	0.272	0.998	---
4	49.00	10.46	U1	Even	0.380	1.741	---

5	49.00	13.80	U1	Odd	0.334	1.176	---
6	49.00	11.53	U1	Odd	0.278	1.045	---

Punching Shear Around Columns:

Units: Vu (kip), Munb (k-ft), vu (psi), Phi*vc (psi)

Supp	Vu	vu	Munb	Comb	Pat	GammaV	vu	Phi*vc
1	36.75	67.5	33.79	U1	Even	0.374	105.0	215.0
2	89.70	107.5	-8.81	U1	S2	0.400	113.8	215.0
3	82.37	98.7	1.82	U1	S3	0.400	100.0	215.0
4	80.24	96.2	-1.68	U1	S4	0.400	97.4	215.0
5	78.25	93.8	-4.03	U1	S5	0.400	96.7	215.0
6	58.04	92.7	15.14	U1	All	0.400	120.7	215.0

Punching Shear Around Drops:

Units: Vu (kip), vu (psi), Phi*vc (psi)

Supp	Vu	Comb	Pat	vu	Phi*vc
1	33.04	U1	S1	31.8	163.4
2	83.67	U1	S2	44.4	148.6
3	76.34	U1	S3	40.5	148.6
4	74.88	U1	S4	41.8	150.8
5	73.77	U1	S5	44.7	154.4
6	53.27	U1	S6	32.9	143.4

Maximum Deflections:

Units: Dz (in)

	Frame			Column Strip			Middle Strip		
Span	Dz(DEAD)	Dz(LIVE)	Dz(TOTAL)	Dz(DEAD)	Dz(LIVE)	Dz(TOTAL)	Dz(DEAD)	Dz(LIVE)	Dz(TOTAL)
1	0.005	0.002	0.007	0.007	0.003	0.011	0.002	0.001	0.003
2	-0.040	-0.018	-0.058	-0.059	-0.027	-0.086	-0.021	-0.010	-0.031
3	-0.016	-0.007	-0.023	-0.021	-0.009	-0.031	-0.010	-0.004	-0.015
4	-0.021	-0.011	-0.032	-0.029	-0.015	-0.043	-0.014	-0.007	-0.021
5	-0.025	-0.006	-0.030	-0.034	-0.008	-0.041	-0.016	-0.004	-0.020
6	0.003	-0.010	-0.008	0.005	-0.020	-0.015	0.001	-0.005	-0.004
7	-0.025	0.008	-0.018	-0.052	0.016	-0.036	-0.008	0.003	-0.006

Material Takeoff:

Reinforcement in the Direction of Analysis

Top Steel: 939.644 lb = 0.591 lb/ft^2
Bottom Steel: 842.702 lb = 0.530 lb/ft^2
Stirrup: 0.000 lb = 0.000 lb/ft^2
Total steel: 1782.346 lb = 1.122 lb/ft^2
Concrete Volume: 1109.009 ft^3 = 0.698 ft^3/ft^2

[3] COLUMN AXIAL FORCES AND MOMENTS

Units: P (kip), M (k-ft)

Supp	Case/Patt	P (axial)	Mb[top]	Ma[bottom]
1	SELF	15.82	-19.45	0.00
	Dead	1.38	-1.93	0.00
	Live-1	6.91	-9.67	0.00
	Live-2	-0.90	3.10	0.00
	Live-3	7.81	-12.77	0.00
	Live-4	7.58	-11.99	0.00
2	SELF	37.67	4.77	0.00
	Dead	3.63	0.47	0.00
	Live-1	18.07	2.45	0.00
	Live-2	9.32	-7.98	0.00
	Live-3	8.75	10.44	0.00
	Live-4	19.10	0.87	0.00
3	SELF	33.50	-1.18	0.00
	Dead	3.21	-0.11	0.00
	Live-1	16.32	-0.95	0.00
	Live-2	8.12	9.02	0.00
	Live-3	8.20	-9.97	0.00
	Live-4	18.56	-0.00	0.00
4	SELF	33.47	0.08	0.00
	Dead	3.24	-0.01	0.00
	Live-1	15.11	1.52	0.00
	Live-2	7.07	-8.67	0.00
	Live-3	8.04	10.20	0.00
	Live-4	17.32	0.93	0.00
5	SELF	25.05	6.15	0.00
	Dead	2.31	0.71	0.00
	Live-1	21.78	-3.60	0.00
	Live-2	7.99	7.88	0.00

	Live-3	13.80	-11.48	0.00
	Live-4	23.84	-3.27	0.00
6	SELF	20.84	-5.06	0.00
	Dead	2.26	-0.84	0.00
	Live-1	15.72	6.94	0.00
	Live-2	3.37	-6.44	0.00
	Live-3	12.35	13.38	0.00
	Live-4	16.73	9.06	0.00
Sum	SELF	166.35	-14.69	0.00
	Dead	16.02	-1.72	0.00
	Live-1	93.92	-3.29	0.00
	Live-2	34.98	-3.09	0.00
	Live-3	58.94	-0.20	0.00

Calculation Results - Graphical Output

The following diagrams illustrate the graphical output for Example 1.

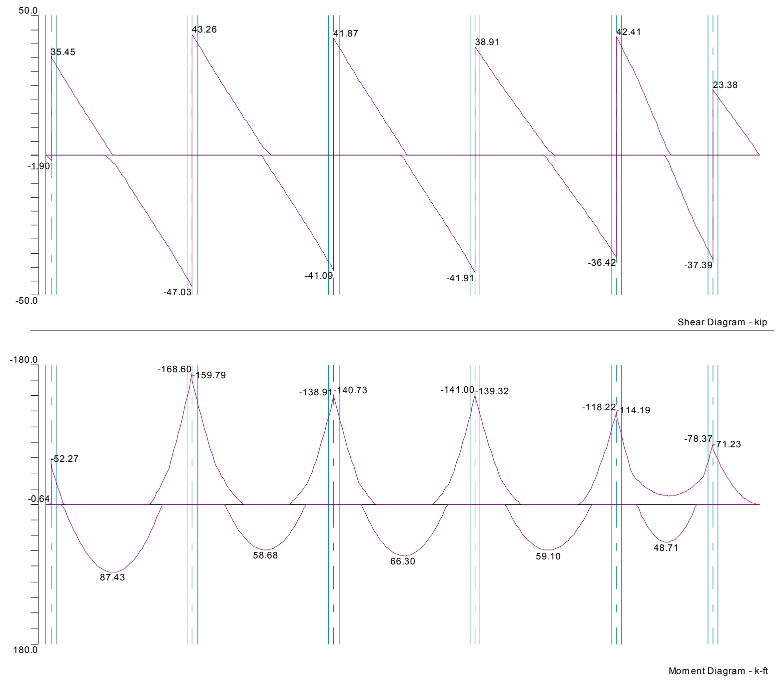


Figure 6-2 Shear and Moment Diagram

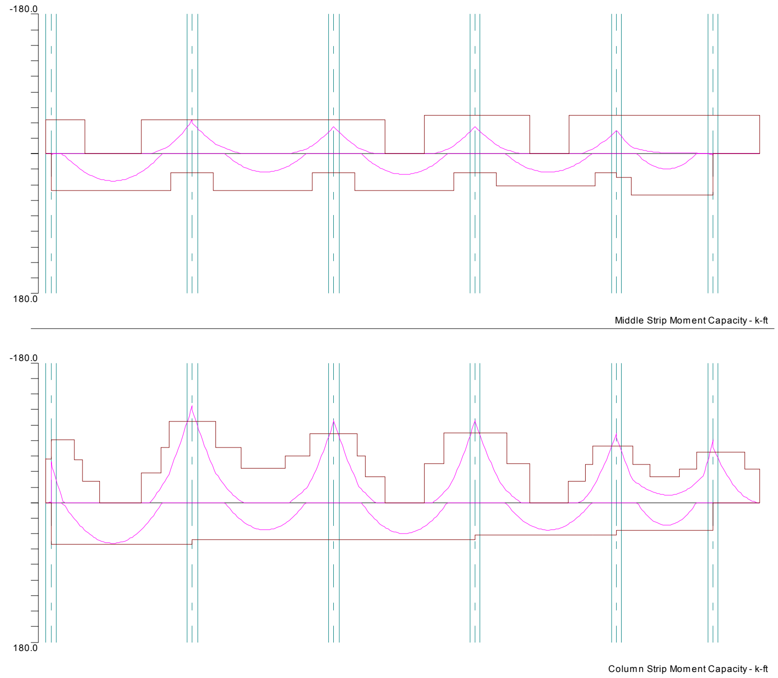


Figure 6-3 Moment Capacity Diagram

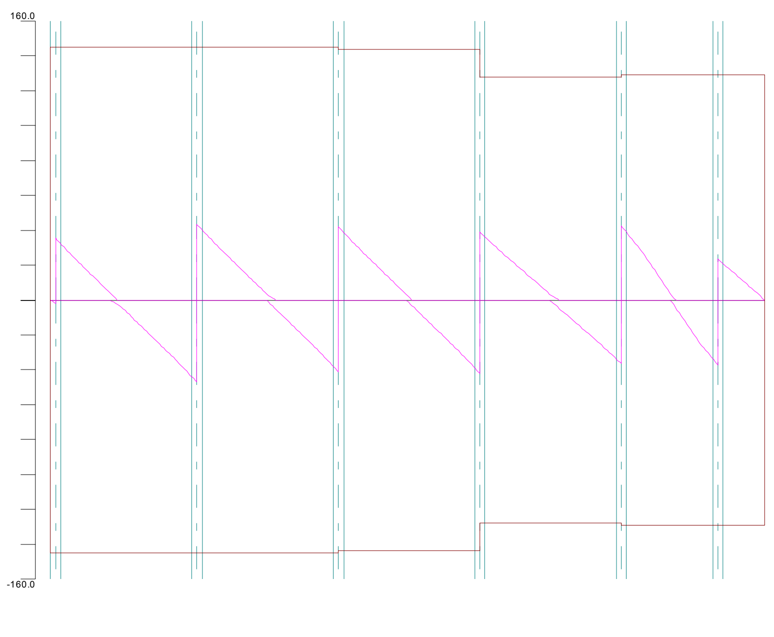


Figure 6-4 Shear Capacity Diagram

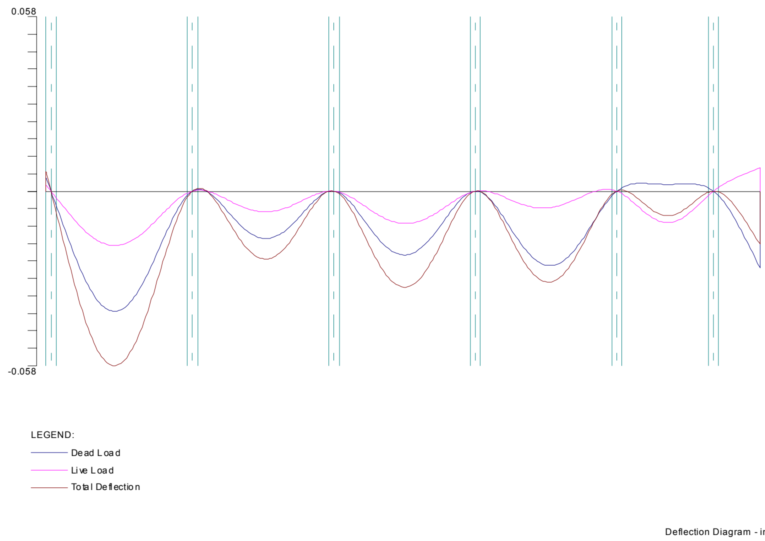


Figure 6-5 Deflection Diagram

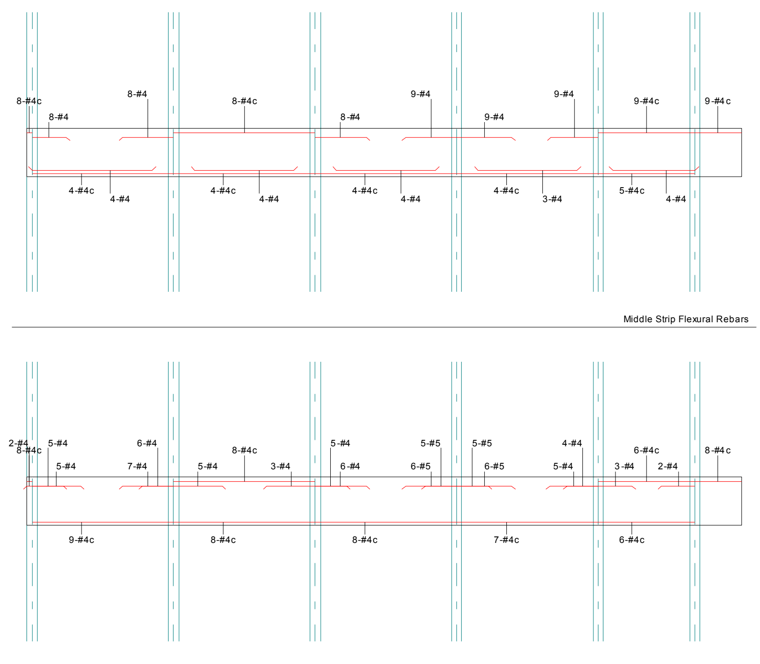


Figure 6-6 Reinforcement Diagram

Beam-Supported Slab

Problem Description

Analyze and design an exterior strip shown in Figure 6-7.

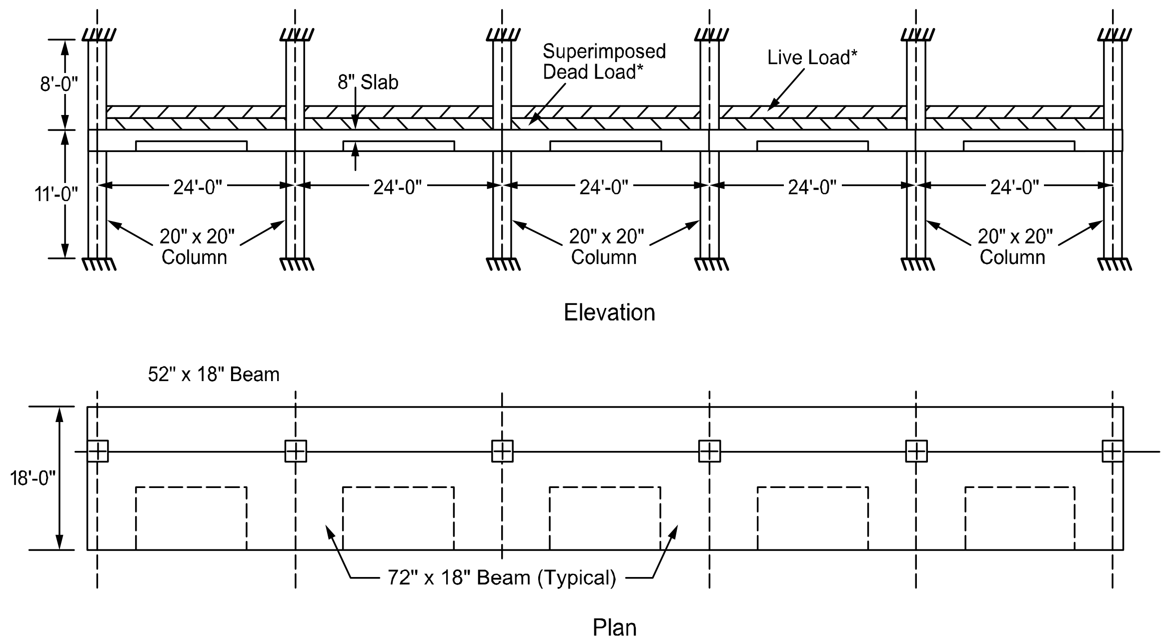


Figure 6-7 Example beam-supported slab

Calculation Results - Text Output

The following listing contains the first three sections of the result report for Example 2.

```
                        ooooooo      oooooo      ooooo
                        oooooooo    oooooooo    ooooooo
                        oo    oo    oo    oo    oo   oo
                        oo    oo    oo          oo   oo
                        oooooooo    oo          ooooooo   ooooo
                        ooooooo     oo    oo    ooooooo   ooooo
                        oo          oo    oo    oo   oo
                        oo          oooooooo    oo   oo
                        oo           oooooo     oo   oo


                      oooooo      o                    o
                     oooooooo    oo         ooooo      oo
                     oo          oo        o    oo     oo
                     oooo        oo        o    oo     oo
                      oooooo     oo         oooooo     oooooo
                         oooo    oo        oo   oo     oo   oo
                           oo    oo        oo   oo     oo   oo
                     oooooooo    oo   o    oo   oo     oo   oo
                      oooooo      ooo       ooooo o     ooooo

      =============================================================================
                                pcaSlab V1.01 (TM)
              A Computer Program for Analysis and Design of Slab Systems
      =============================================================================
                    Copyright © 2000-2003, Portland Cement Association
                                All rights reserved

          Licensee stated above acknowledges that Portland Cement Association
         (PCA) is not and  cannot  be  responsible for  either the  accuracy  or
         adequacy of  the material supplied  as input  for processing   by  the
         pcaSlab computer program.  Furthermore, PCA neither makes any warranty
         expressed nor implied with respect  to  the  correctness of the  output
         prepared by the  pcaSlab  program.  Although  PCA  has  endeavored  to
         produce pcaSlab error free the  program is not and cannot be certified
         infallible.  The final and only responsibility for analysis, design and
         engineering documents is the licensees. Accordingly, PCA disclaims  all
         responsibility in contract, negligence or other tort for any  analysis,
         design  or engineering documents prepared in connection with the use of
         the pcaSlab program.


==============================================================================================
[1] INPUT ECHO
==============================================================================================

General Information:
====================
   File name: C:\Program Files\PCA\pcaslab\Examples Adoss\pltbms.slb
   Project: FLAT PLATE WITH BEAMS
   Frame: Example 4                                Engineer: PCA
   Code: ACI 318-99       Mode: Design             Rebar Database: ASTM A615
   Number of supports = 6 + Left cantilever + Right cantilever

   Live load pattern ratio = 75%
   Minimum free edge for punching shear = 10 times slab thickness
   Deflections are based on cracked section properties.

Material Properties:
====================
            Slabs|Beams       Columns
           ------------   ------------
   wc     =         150            150 lb/ft3
   f'c    =           4              4 ksi
   Ec     =     3834.25        3834.25 ksi
   fr     =      0.4743         0.4743 ksi

   fy     =          60 ksi, Bars are not epoxy-coated
   fyv    =          60 ksi
   Es     =       29000 ksi

Rebar Database:
===============
   Units: Db (in), Ab (in^2), Wb (lb/ft)
   Size       Db       Ab       Wb    Size       Db       Ab       Wb
   ---- -------- -------- --------    ---- -------- -------- --------
     #3     0.38     0.11     0.38      #4     0.50     0.20     0.67
     #5     0.63     0.31     1.04      #6     0.75     0.44     1.50
     #7     0.88     0.60     2.04      #8     1.00     0.79     2.67
     #9     1.13     1.00     3.40     #10     1.27     1.27     4.30
    #11     1.41     1.56     5.31     #14     1.69     2.25     7.65
```

```
    #18     2.26     4.00    13.60

Span Data:
==========
   Slabs: L1, wL, wR (ft); t (in)
   Span Loc        L1        t       wL       wR     Hmin
   ---- ---- -------- -------- -------- -------- --------
      1 ExtL    0.833     8.00    2.167   10.000     4.87 LC
      2 ExtL   24.000     8.00    2.167   10.000     5.35
      3 ExtL   24.000     8.00    2.167   10.000     5.35
      4 ExtL   24.000     8.00    2.167   10.000     5.35
      5 ExtL   24.000     8.00    2.167   10.000     5.35
      6 ExtL   24.000     8.00    2.167   10.000     5.35
      7 ExtL    0.833     8.00    2.167   10.000     4.87 RC

   Ribs and Longitudinal Beams: b, h, Sp (in)
        __________Ribs___________     _______Beams_____
   Span        b        h       Sp          b        h
   ---- -------- -------- --------     -------- --------
      1     0.00     0.00     0.00        52.00    18.00
      2     0.00     0.00     0.00        52.00    18.00
      3     0.00     0.00     0.00        52.00    18.00
      4     0.00     0.00     0.00        52.00    18.00
      5     0.00     0.00     0.00        52.00    18.00
      6     0.00     0.00     0.00        52.00    18.00
      7     0.00     0.00     0.00        52.00    18.00

Support Data:
=============
   Columns: c1a, c2a, c1b, c2b (in); Ha, Hb (ft)
   Supp      c1a      c2a       Ha        c1b      c2b       Hb  Red%
   ---- -------- -------- --------   -------- -------- -------- ----
      1    20.00    20.00    8.000      20.00    20.00   11.000   100
      2    20.00    20.00    8.000      20.00    20.00   11.000   100
      3    20.00    20.00    8.000      20.00    20.00   11.000   100
      4    20.00    20.00    8.000      20.00    20.00   11.000   100
      5    20.00    20.00    8.000      20.00    20.00   11.000   100
      6    20.00    20.00    8.000      20.00    20.00   11.000   100

   Transverse Beams: b, h, Ecc (in)
   Supp        b        h      Ecc
   ---- -------- -------- --------
      1    72.00    18.00    26.00
      2    72.00    18.00     0.00
      3    72.00    18.00     0.00
      4    72.00    18.00     0.00
      5    72.00    18.00     0.00
      6    72.00    18.00   -26.00

Load Data:
==========
   Load Cases and Combinations:
   Case     SELF     Dead     Live
   Type     DEAD     DEAD     LIVE
   U1      1.400    1.400    1.700

   Span Loads:
   Span Case                Wa            La            Wb           Lb
   ---- -------- ------------ ------------ ------------ ------------
   Line Loads - Wa | Wb (lb/ft), La | Lb (ft):
      6 Live               500           2.5           500           10
   Area Loads - Wa (lb/ft2):
      1 Dead                10
      2 Dead                10
      3 Dead                10
      4 Dead                10
      5 Dead                10
      6 Dead                10
      7 Dead                10
      1 Live               100
      2 Live               100
      3 Live               100
      4 Live               100
      5 Live               100
      6 Live               100
      7 Live               100

Reinforcement Criteria:
=======================
                 _____Top bars___  ___Bottom bars__  ____Stirrups____
                   Min      Max      Min      Max      Min      Max
                 -------  -------  -------  -------  -------  -------
   Slabs and Ribs:
     Bar Size         #4       #8       #4       #8
     Bar spacing    6.00    18.00     6.00    18.00 in
     Reinf ratio    0.18     2.00     0.18     2.00 %
     Cover          1.20              1.20          in

   Beams:
     Bar Size         #4      #14       #4      #14       #3       #5
     Bar spacing    1.00    18.00     1.00    18.00     6.00    18.00 in
     Reinf ratio    0.20     2.00     0.20     2.00 %
```

Cover 1.20 1.20 in

==
[2] DESIGN RESULTS
==

Top Reinforcement:
==================

Units: Width (ft), Mmax (k-ft), Xmax (ft), As (in^2), Sp (in)

Span	Strip	Zone	Width	Mmax	Xmax	AsMin	AsMax	SpReq	AsReq	Bars
1	Column	Left	2.83	0.00	0.000	0.490	0.000	11.333	0.000	3-#4
		Middle	2.83	0.00	0.000	0.490	0.000	0.000	0.000	3-#4
		Right	2.83	0.00	0.000	0.490	4.454	11.333	0.000	3-#4
	Middle	Left	5.00	0.00	0.000	0.864	7.860	12.000	0.000	5-#4
		Middle	5.00	0.00	0.000	0.864	7.860	0.000	0.000	5-#4
		Right	5.00	0.00	0.000	0.864	7.860	12.000	0.000	5-#4
	Beam	Left	4.33	0.00	0.000	1.721	17.212	6.008	0.000	9-#4
		Middle	4.33	0.00	0.000	1.721	17.212	0.000	0.000	9-#4
		Right	4.33	0.00	0.000	1.721	17.212	6.008	0.000	9-#4
2	Column	Left	2.83	17.09	0.833	0.490	4.454	11.333	0.594	3-#4
		Middle	2.83	0.00	0.000	0.000	0.000	0.000	0.000	---
		Right	2.83	32.43	23.167	0.490	4.412	8.500	1.165	4-#5
	Middle	Left	5.00	12.95	0.833	0.864	7.860	12.000	0.444	5-#4
		Middle	5.00	0.00	0.000	0.000	0.000	0.000	0.000	---
		Right	5.00	24.58	23.167	0.864	7.860	12.000	0.850	5-#4
	Beam	Left	4.33	96.82	0.833	1.753	17.212	6.008	1.318	9-#4
		Middle	4.33	0.00	0.000	0.000	0.000	0.000	0.000	---
		Right	4.33	183.79	23.167	2.869	17.212	3.433	2.534	15-#4
3	Column	Left	2.83	29.24	0.833	0.490	4.412	8.500	1.045	4-#5
		Middle	2.83	0.00	0.000	0.000	0.000	0.000	0.000	---
		Right	2.83	26.77	23.167	0.490	4.454	6.800	0.943	5-#4
	Middle	Left	5.00	22.16	0.833	0.864	7.860	12.000	0.765	5-#4
		Middle	5.00	0.00	0.000	0.000	0.000	0.000	0.000	---
		Right	5.00	20.29	23.167	0.864	7.860	12.000	0.699	5-#4
	Beam	Left	4.33	165.71	0.833	2.869	17.212	3.433	2.278	15-#4
		Middle	4.33	0.00	0.000	0.000	0.000	0.000	0.000	---
		Right	4.33	151.69	23.167	2.768	17.212	3.433	2.081	15-#4
4	Column	Left	2.83	27.37	0.833	0.490	4.454	6.800	0.966	5-#4
		Middle	2.83	0.00	0.000	0.000	0.000	0.000	0.000	---
		Right	2.83	27.19	23.167	0.490	4.454	6.800	0.959	5-#4
	Middle	Left	5.00	20.74	0.833	0.864	7.860	12.000	0.715	5-#4
		Middle	5.00	0.00	0.000	0.000	0.000	0.000	0.000	---
		Right	5.00	20.61	23.167	0.864	7.860	12.000	0.711	5-#4
	Beam	Left	4.33	155.10	0.833	2.832	17.212	3.433	2.129	15-#4
		Middle	4.33	0.00	0.000	0.000	0.000	0.000	0.000	---
		Right	4.33	154.09	23.167	2.813	17.212	3.433	2.115	15-#4
5	Column	Left	2.83	26.40	0.833	0.490	4.454	6.800	0.930	5-#4
		Middle	2.83	0.00	0.000	0.000	0.000	0.000	0.000	---
		Right	2.83	30.05	23.167	0.490	4.412	8.500	1.076	4-#5
	Middle	Left	5.00	20.01	0.833	0.864	7.860	12.000	0.690	5-#4
		Middle	5.00	0.00	0.000	0.000	0.000	0.000	0.000	---
		Right	5.00	22.78	23.167	0.864	7.860	12.000	0.787	5-#4
	Beam	Left	4.33	149.62	0.833	2.729	17.212	3.433	2.052	15-#4
		Middle	4.33	0.00	0.000	0.000	0.000	0.000	0.000	---
		Right	4.33	170.30	23.167	2.869	17.212	3.433	2.343	15-#4
6	Column	Left	2.83	34.35	0.833	0.490	4.412	8.500	1.238	4-#5
		Middle	2.83	0.00	0.000	0.000	0.000	0.000	0.000	---
		Right	2.83	18.00	23.167	0.490	4.454	8.500	0.626	4-#4
	Middle	Left	5.00	26.04	0.833	0.864	7.860	12.000	0.902	5-#4
		Middle	5.00	0.00	0.000	0.000	0.000	0.000	0.000	---
		Right	5.00	13.64	23.167	0.864	7.860	12.000	0.468	5-#4
	Beam	Left	4.33	194.66	0.833	2.869	17.212	3.433	2.688	15-#4
		Middle	4.33	0.00	0.000	0.000	0.000	0.000	0.000	---
		Right	4.33	101.98	23.167	1.848	17.212	5.340	1.389	10-#4
7	Column	Left	2.83	0.00	0.833	0.490	4.454	8.500	0.000	4-#4
		Middle	2.83	0.00	0.833	0.490	0.000	0.000	0.000	3-#4
		Right	2.83	0.00	0.833	0.490	0.000	11.333	0.000	3-#4
	Middle	Left	5.00	0.00	0.833	0.864	7.860	12.000	0.000	5-#4
		Middle	5.00	0.00	0.833	0.864	7.860	0.000	0.000	5-#4
		Right	5.00	0.00	0.833	0.864	7.860	12.000	0.000	5-#4

Beam	Left	4.33	0.00	0.833	1.721	17.212	5.340	0.000	10-#4
	Middle	4.33	0.00	0.833	1.721	17.212	0.000	0.000	9-#4
	Right	4.33	0.00	0.833	1.721	17.212	6.008	0.000	9-#4

Top Bar Details:
================

Units: Length (ft)

		Left				Continuous		Right			
Span	Strip	Bars	Length	Bars	Length	Bars	Length	Bars	Length	Bars	Length
1	Column	---		---		3-#4	0.83	---		---	
	Middle	---		---		5-#4	0.83	---		---	
	Beam	---		---		9-#4	0.83	---		---	
2	Column	2-#4	8.20	1-#4	5.30	---		2-#5	8.35	2-#5	5.30
	Middle	5-#4	5.75	---		---		5-#4	8.35	---	
	Beam	5-#4	5.36	4-#4	2.81	---		8-#4	8.35	7-#4	3.76
3	Column	2-#5	8.65	2-#5	5.30	---		3-#4	8.20	2-#4	5.30
	Middle	5-#4	8.65	---		---		5-#4	8.15	---	
	Beam	8-#4	8.65	7-#4	3.42	---		8-#4	8.15	7-#4	3.08
4	Column	3-#4	8.20	2-#4	5.30	---		3-#4	8.20	2-#4	5.30
	Middle	5-#4	8.15	---		---		5-#4	8.15	---	
	Beam	8-#4	8.15	7-#4	3.16	---		8-#4	8.15	7-#4	3.13
5	Column	3-#4	8.20	2-#4	5.30	---		2-#5	8.65	2-#5	5.30
	Middle	5-#4	8.15	---		---		5-#4	8.65	---	
	Beam	8-#4	8.15	7-#4	3.03	---		8-#4	8.65	7-#4	3.53
6	Column	2-#5	8.35	2-#5	5.30	---		2-#4	8.20	2-#4	5.30
	Middle	5-#4	8.35	---		---		5-#4	5.75	---	
	Beam	8-#4	8.35	7-#4	3.87	---		5-#4	5.36	5-#4	2.93
7	Column	1-#4	0.83	---		3-#4	0.83	---		---	
	Middle	---		---		5-#4	0.83	---		---	
	Beam	1-#4	0.83	---		9-#4	0.83	---		---	

Bottom Reinforcement:
=====================

Units: Width (ft), Mmax (k-ft), Xmax (ft), As (in^2), Sp (in)

Span	Strip	Width	Mmax	Xmax	AsMin	AsMax	SpReq	AsReq	Bars
1	Column	2.83	0.00	0.000	0.000	4.454	0.000	0.000	---
	Middle	5.00	0.00	0.000	0.000	7.860	0.000	0.000	---
	Beam	4.33	0.00	0.000	0.000	17.212	0.000	0.000	---
2	Column	2.83	16.23	10.857	0.490	4.454	11.333	0.563	3-#4
	Middle	5.00	12.30	10.857	0.864	7.860	12.000	0.421	5-#4
	Beam	4.33	91.99	10.857	1.721	17.212	6.008	1.251	9-#4
3	Column	2.83	12.40	12.250	0.490	4.454	11.333	0.428	3-#4
	Middle	5.00	9.40	12.250	0.864	7.860	12.000	0.321	5-#4
	Beam	4.33	70.26	12.250	1.721	17.212	6.008	0.953	9-#4
4	Column	2.83	12.97	12.000	0.490	4.454	11.333	0.448	3-#4
	Middle	5.00	9.83	12.000	0.864	7.860	12.000	0.336	5-#4
	Beam	4.33	73.49	12.000	1.721	17.212	6.008	0.997	9-#4
5	Column	2.83	12.40	11.750	0.490	4.454	11.333	0.428	3-#4
	Middle	5.00	9.40	11.750	0.864	7.860	12.000	0.321	5-#4
	Beam	4.33	70.26	11.750	1.721	17.212	6.008	0.953	9-#4
6	Column	2.83	16.97	12.896	0.490	4.454	11.333	0.590	3-#4
	Middle	5.00	12.86	12.896	0.864	7.860	12.000	0.441	5-#4
	Beam	4.33	96.18	12.896	1.741	17.212	6.008	1.309	9-#4
7	Column	2.83	0.00	0.833	0.000	4.454	0.000	0.000	---
	Middle	5.00	0.00	0.833	0.000	7.860	0.000	0.000	---
	Beam	4.33	0.00	0.833	0.000	17.212	0.000	0.000	---

Bottom Bar Details:
===================

Units: Start (ft), Length (ft)

		Long Bars			Short Bars		
Span	Strip	Bars	Start	Length	Bars	Start	Length
1	Column	---			---		
	Middle	---			---		
	Beam	---			---		
2	Column	3-#4	0.00	24.00	---		
	Middle	3-#4	0.00	24.00	2-#4	0.00	20.40
	Beam	5-#4	0.00	24.00	4-#4	0.00	15.31
3	Column	3-#4	0.00	24.00	---		
	Middle	3-#4	0.00	24.00	2-#4	3.60	16.80
	Beam	5-#4	0.00	24.00	4-#4	10.85	2.80
4	Column	3-#4	0.00	24.00	---		
	Middle	3-#4	0.00	24.00	2-#4	3.60	16.80
	Beam	5-#4	0.00	24.00	4-#4	10.60	2.80

5	Column	3-#4	0.00	24.00	---		
	Middle	3-#4	0.00	24.00	2-#4	3.60	16.80
	Beam	5-#4	0.00	24.00	4-#4	10.35	2.80
6	Column	3-#4	0.00	24.00	---		
	Middle	3-#4	0.00	24.00	2-#4	3.60	20.40
	Beam	5-#4	0.00	24.00	4-#4	8.08	15.92
7	Column	---			---		
	Middle	---			---		
	Beam	---			---		

Flexural Capacity:
==================

Units: From, To (ft), As (in^2), PhiMn (k-ft)

Span	Strip	From	To	AsTop	AsBot	PhiMn-	PhiMn+
1	Column	0.000	0.833	0.60	0.00	-17.26	0.00
	Middle	0.000	0.833	1.00	0.00	-28.81	0.00
	Beam	0.000	0.833	1.80	0.00	-131.58	0.00
2	Column	0.000	0.833	0.60	0.60	-17.26	17.26
		0.833	5.301	0.60	0.60	-17.26	17.26
		5.301	8.204	0.40	0.60	-11.60	17.26
		8.204	8.650	0.00	0.60	0.00	17.26
		8.650	15.350	0.00	0.60	0.00	17.26
		15.350	15.646	0.00	0.60	0.00	17.26
		15.646	18.699	0.62	0.60	-17.65	17.26
		18.699	23.167	1.24	0.60	-34.40	17.26
		23.167	24.000	1.24	0.60	-34.40	17.26
	Middle	0.000	0.833	1.00	1.00	-28.81	28.81
		0.833	5.747	1.00	1.00	-28.81	28.81
		5.747	8.650	0.00	1.00	0.00	28.81
		8.650	15.350	0.00	1.00	0.00	28.81
		15.350	15.646	0.00	1.00	0.00	28.81
		15.646	20.400	1.00	1.00	-28.81	28.81
		20.400	23.167	1.00	0.60	-28.81	17.45
		23.167	24.000	1.00	0.60	-28.81	17.45
	Beam	0.000	0.833	1.80	1.80	-131.58	131.58
		0.833	2.806	1.80	1.80	-131.58	131.58
		2.806	5.359	1.00	1.80	-73.71	131.58
		5.359	8.650	0.00	1.80	0.00	131.58
		8.650	15.313	0.00	1.80	0.00	131.58
		15.313	15.350	0.00	1.00	0.00	73.71
		15.350	15.646	0.00	1.00	0.00	73.71
		15.646	20.237	1.60	1.00	-117.21	73.71
		20.237	23.167	3.00	1.00	-216.55	73.71
		23.167	24.000	3.00	1.00	-216.55	73.71
3	Column	0.000	0.833	1.24	0.60	-34.40	17.26
		0.833	5.301	1.24	0.60	-34.40	17.26
		5.301	8.646	0.62	0.60	-17.65	17.26
		8.646	8.650	0.00	0.60	0.00	17.26
		8.650	15.350	0.00	0.60	0.00	17.26
		15.350	15.796	0.00	0.60	0.00	17.26
		15.796	18.699	0.60	0.60	-17.26	17.26
		18.699	23.167	1.00	0.60	-28.31	17.26
		23.167	24.000	1.00	0.60	-28.31	17.26
	Middle	0.000	0.833	1.00	0.60	-28.81	17.45
		0.833	3.600	1.00	0.60	-28.81	17.45
		3.600	8.646	1.00	1.00	-28.81	28.81
		8.646	8.650	0.00	1.00	0.00	28.81
		8.650	15.350	0.00	1.00	0.00	28.81
		15.350	15.854	0.00	1.00	0.00	28.81
		15.854	20.400	1.00	1.00	-28.81	28.81
		20.400	23.167	1.00	0.60	-28.81	17.45
		23.167	24.000	1.00	0.60	-28.81	17.45
	Beam	0.000	0.833	3.00	1.00	-216.55	73.71
		0.833	3.421	3.00	1.00	-216.55	73.71
		3.421	8.646	1.60	1.00	-117.21	73.71
		8.646	8.650	0.00	1.00	0.00	73.71
		8.650	10.849	0.00	1.00	0.00	73.71
		10.849	13.651	0.00	1.80	0.00	131.58
		13.651	15.350	0.00	1.00	0.00	73.71
		15.350	15.854	0.00	1.00	0.00	73.71
		15.854	20.918	1.60	1.00	-117.21	73.71
		20.918	23.167	3.00	1.00	-216.55	73.71
		23.167	24.000	3.00	1.00	-216.55	73.71
4	Column	0.000	0.833	1.00	0.60	-28.31	17.26
		0.833	5.301	1.00	0.60	-28.31	17.26
		5.301	8.204	0.60	0.60	-17.26	17.26
		8.204	8.650	0.00	0.60	0.00	17.26
		8.650	15.350	0.00	0.60	0.00	17.26
		15.350	15.796	0.00	0.60	0.00	17.26
		15.796	18.699	0.60	0.60	-17.26	17.26
		18.699	23.167	1.00	0.60	-28.31	17.26
		23.167	24.000	1.00	0.60	-28.31	17.26
	Middle	0.000	0.833	1.00	0.60	-28.81	17.45
		0.833	3.600	1.00	0.60	-28.81	17.45
		3.600	8.146	1.00	1.00	-28.81	28.81

		8.146	8.650	0.00	1.00	0.00	28.81
		8.650	15.350	0.00	1.00	0.00	28.81
		15.350	15.854	0.00	1.00	0.00	28.81
		15.854	20.400	1.00	1.00	-28.81	28.81
		20.400	23.167	1.00	0.60	-28.81	17.45
		23.167	24.000	1.00	0.60	-28.81	17.45
	Beam	0.000	0.833	3.00	1.00	-216.55	73.71
		0.833	3.156	3.00	1.00	-216.55	73.71
		3.156	8.146	1.60	1.00	-117.21	73.71
		8.146	8.650	0.00	1.00	0.00	73.71
		8.650	10.599	0.00	1.00	0.00	73.71
		10.599	13.401	0.00	1.80	0.00	131.58
		13.401	15.350	0.00	1.00	0.00	73.71
		15.350	15.854	0.00	1.00	0.00	73.71
		15.854	20.868	1.60	1.00	-117.21	73.71
		20.868	23.167	3.00	1.00	-216.55	73.71
		23.167	24.000	3.00	1.00	-216.55	73.71
5	Column	0.000	0.833	1.00	0.60	-28.31	17.26
		0.833	5.301	1.00	0.60	-28.31	17.26
		5.301	8.204	0.60	0.60	-17.26	17.26
		8.204	8.650	0.00	0.60	0.00	17.26
		8.650	15.350	0.00	0.60	0.00	17.26
		15.350	15.354	0.00	0.60	0.00	17.26
		15.354	18.699	0.62	0.60	-17.65	17.26
		18.699	23.167	1.24	0.60	-34.40	17.26
		23.167	24.000	1.24	0.60	-34.40	17.26
	Middle	0.000	0.833	1.00	0.60	-28.81	17.45
		0.833	3.600	1.00	0.60	-28.81	17.45
		3.600	8.146	1.00	1.00	-28.81	28.81
		8.146	8.650	0.00	1.00	0.00	28.81
		8.650	15.350	0.00	1.00	0.00	28.81
		15.350	15.354	0.00	1.00	0.00	28.81
		15.354	20.400	1.00	1.00	-28.81	28.81
		20.400	23.167	1.00	0.60	-28.81	17.45
		23.167	24.000	1.00	0.60	-28.81	17.45
	Beam	0.000	0.833	3.00	1.00	-216.55	73.71
		0.833	3.034	3.00	1.00	-216.55	73.71
		3.034	8.146	1.60	1.00	-117.21	73.71
		8.146	8.650	0.00	1.00	0.00	73.71
		8.650	10.349	0.00	1.00	0.00	73.71
		10.349	13.151	0.00	1.80	0.00	131.58
		13.151	15.350	0.00	1.00	0.00	73.71
		15.350	15.354	0.00	1.00	0.00	73.71
		15.354	20.466	1.60	1.00	-117.21	73.71
		20.466	23.167	3.00	1.00	-216.55	73.71
		23.167	24.000	3.00	1.00	-216.55	73.71
6	Column	0.000	0.833	1.24	0.60	-34.40	17.26
		0.833	5.301	1.24	0.60	-34.40	17.26
		5.301	8.354	0.62	0.60	-17.65	17.26
		8.354	8.650	0.00	0.60	0.00	17.26
		8.650	15.350	0.00	0.60	0.00	17.26
		15.350	15.796	0.00	0.60	0.00	17.26
		15.796	18.699	0.40	0.60	-11.60	17.26
		18.699	23.167	0.80	0.60	-22.83	17.26
		23.167	24.000	0.80	0.60	-22.83	17.26
	Middle	0.000	0.833	1.00	0.60	-28.81	17.45
		0.833	3.600	1.00	0.60	-28.81	17.45
		3.600	8.354	1.00	1.00	-28.81	28.81
		8.354	8.650	0.00	1.00	0.00	28.81
		8.650	15.350	0.00	1.00	0.00	28.81
		15.350	18.253	0.00	1.00	0.00	28.81
		18.253	23.167	1.00	1.00	-28.81	28.81
		23.167	24.000	1.00	1.00	-28.81	28.81
	Beam	0.000	0.833	3.00	1.00	-216.55	73.71
		0.833	3.873	3.00	1.00	-216.55	73.71
		3.873	8.079	1.60	1.00	-117.21	73.71
		8.079	8.354	1.60	1.80	-117.21	131.58
		8.354	8.650	0.00	1.80	0.00	131.58
		8.650	15.350	0.00	1.80	0.00	131.58
		15.350	18.641	0.00	1.80	0.00	131.58
		18.641	21.074	1.00	1.80	-73.71	131.58
		21.074	23.167	2.00	1.80	-145.90	131.58
		23.167	24.000	2.00	1.80	-145.90	131.58
7	Column	0.000	0.833	0.80	0.00	-22.83	0.00
		0.833	0.833	0.60	0.00	-17.26	0.00
	Middle	0.000	0.833	1.00	0.00	-28.81	0.00
		0.833	0.833	1.00	0.00	-28.81	0.00
	Beam	0.000	0.833	2.00	0.00	-145.90	0.00
		0.833	0.833	1.80	0.00	-131.58	0.00

Longitudinal Beam Shear Reinforcement Required:
===

Units: d (in), Start, End (ft), PhiVc, Vu (kip), Av/s (in^2/in)

Span	d	PhiVc	Start	End	Vu	Av/s
1	16.55	92.53	0.000	0.000	0.00	0.000
2	16.55	92.53	2.212	5.009	44.52	0.000
			5.009	7.805	27.54	0.000
			7.805	10.602	14.18	0.000

			10.602	13.398	12.11	0.000
			13.398	16.195	25.25	0.000
			16.195	18.991	38.39	0.000
			18.991	21.788	52.62	0.043
3	16.55	92.53	2.212	5.009	47.91	0.043
			5.009	7.805	33.68	0.000
			7.805	10.602	20.54	0.000
			10.602	13.398	7.85	0.000
			13.398	16.195	18.89	0.000
			16.195	18.991	32.04	0.000
			18.991	21.788	46.26	0.000
4	16.55	92.53	2.212	5.009	47.14	0.043
			5.009	7.805	32.92	0.000
			7.805	10.602	19.78	0.000
			10.602	13.398	7.29	0.000
			13.398	16.195	19.66	0.000
			16.195	18.991	32.80	0.000
			18.991	21.788	47.03	0.043
5	16.55	92.53	2.212	5.009	45.87	0.000
			5.009	7.805	31.65	0.000
			7.805	10.602	18.53	0.000
			10.602	13.398	8.14	0.000
			13.398	16.195	20.93	0.000
			16.195	18.991	34.07	0.000
			18.991	21.788	48.30	0.043
6	16.55	92.53	2.212	5.009	57.78	0.043
			5.009	7.805	41.43	0.000
			7.805	10.602	25.90	0.000
			10.602	13.398	10.89	0.000
			13.398	16.195	15.39	0.000
			16.195	18.991	28.76	0.000
			18.991	21.788	45.73	0.000
7	16.55	92.53	0.833	0.833	0.00	0.000

Longitudinal Beam Shear Reinforcement Details:
===

Units: spacing & distance (in).

Span	Size	Stirrups (2 legs each unless otherwise noted)
1	#3	--- None ---
2	#3	<-- 214.9 --> + 7 @ 7.5 [3L]
3	#3	7 @ 7.5 [3L] + <-- 214.9 -->
4	#3	7 @ 7.5 [3L] + <-- 167.8 --> + 7 @ 7.5 [3L]
5	#3	<-- 214.9 --> + 7 @ 7.5 [3L]
6	#3	7 @ 7.5 [3L] + <-- 214.9 -->
7	#3	--- None ---

Beam Shear Capacity:
====================

Units: d, Sp (in), Start, End (ft), PhiVc, PhiVn, Vmax (kip), Av (in^2)

Span	d	PhiVc	Start	End	Av	Sp	PhiVn	Vmax
1	16.55	92.53	0.000	0.833	0.000	-----	46.26	5.06
2	16.55	92.53	0.000	1.083	0.000	-----	46.26	57.95
			1.083	18.991	0.000	-----	46.26	51.38
			18.991	23.366	0.330	7.5	129.67	62.20
			23.366	24.000	0.330	7.5	129.67	66.05
3	16.55	92.53	0.000	1.083	0.330	7.5	129.67	61.34
			1.083	5.458	0.330	7.5	129.67	54.76
			5.458	23.366	0.000	-----	46.26	55.85
			23.366	24.000	0.000	-----	46.26	59.69
4	16.55	92.53	0.000	1.083	0.330	7.5	129.67	60.58
			1.083	5.458	0.330	7.5	129.67	54.00
			5.458	19.440	0.000	-----	46.26	34.91
			19.440	23.815	0.330	7.5	129.67	59.34
			23.815	24.000	0.330	7.5	129.67	60.46
5	16.55	92.53	0.000	1.083	0.000	-----	46.26	59.30
			1.083	18.991	0.000	-----	46.26	52.73
			18.991	23.366	0.330	7.5	129.67	57.88
			23.366	24.000	0.330	7.5	129.67	61.73
6	16.55	92.53	0.000	1.083	0.330	7.5	129.67	71.21
			1.083	5.458	0.330	7.5	129.67	64.64
			5.458	23.366	0.000	-----	46.26	55.31
			23.366	24.000	0.000	-----	46.26	59.16
7	16.55	92.53	0.000	0.833	0.000	-----	46.26	5.06

Slab Shear Capacity:
====================

Units: b, d (in), Xu (ft), PhiVc, Vu(kip)

Span	b	d	Vratio	PhiVc	Vu	Xu
1	94.00	6.55	0.000	66.20	0.00	0.00
2	94.00	6.49	0.000	65.57	0.00	22.63
3	94.00	6.49	0.000	65.57	0.00	1.37
4	94.00	6.55	0.000	66.20	0.00	1.38
5	94.00	6.49	0.000	65.57	0.00	22.63
6	94.00	6.49	0.000	65.57	0.00	1.37
7	94.00	6.55	0.000	66.20	0.00	0.00

Flexural Transfer of Negative Unbalanced Moment at Supports:
==

Units: Width (in), Munb (k-ft), As (in^2)

Supp	Width	Munb	Comb	Pat	AsReq	AsProv	Additional Bars
1	44.00	105.16	U1	All	4.077	0.600	18-#4
2	44.00	43.65	U1	Even	1.572	1.240	2-#5
3	44.00	35.01	U1	Even	1.235	1.000	2-#4
4	44.00	35.74	U1	Even	1.261	1.000	2-#4
5	44.00	49.15	U1	Even	1.782	1.240	2-#5
6	44.00	109.93	U1	All	4.294	0.800	18-#4

Punching Shear Around Columns:
==============================

Units: Vu (kip), Munb (k-ft), vu (psi), Phi*vc (psi)

Supp	Vu	vu	Munb	Comb	Pat	GammaV	vu	Phi*vc
1	61.09	75.8	131.58	U1	All	0.385	139.0	215.0
2	125.24	97.4	-27.60	U1	All	0.400	106.9	215.0
3	118.12	91.8	5.21	U1	All	0.400	93.6	215.0
4	117.61	91.5	-6.81	U1	All	0.400	93.8	215.0
5	130.79	101.7	39.82	U1	All	0.400	115.5	215.0
6	62.30	77.3	-138.55	U1	All	0.385	143.8	215.0

Maximum Deflections:
====================

Units: Dz (in)

	Frame			Column Strip			Middle Strip		
Span	Dz(DEAD)	Dz(LIVE)	Dz(TOTAL)	Dz(DEAD)	Dz(LIVE)	Dz(TOTAL)	Dz(DEAD)	Dz(LIVE)	Dz(TOTAL)
1	0.002	0.001	0.003	0.002	0.001	0.003	0.007	0.004	0.011
2	-0.024	-0.015	-0.039	-0.027	-0.016	-0.043	-0.038	-0.023	-0.061
3	-0.015	-0.010	-0.025	-0.017	-0.011	-0.027	-0.024	-0.015	-0.039
4	-0.016	-0.010	-0.027	-0.018	-0.012	-0.030	-0.026	-0.017	-0.042
5	-0.015	-0.009	-0.024	-0.017	-0.010	-0.027	-0.024	-0.014	-0.038
6	-0.024	-0.017	-0.041	-0.027	-0.019	-0.046	-0.038	-0.027	-0.065
7	0.002	0.001	0.003	0.002	0.001	0.003	0.007	0.004	0.011

Material Takeoff:
=================

Reinforcement in the Direction of Analysis

Top Steel: 1057.708 lb = 0.715 lb/ft^2
Bottom Steel: 1109.521 lb = 0.750 lb/ft^2
Stirrup: 772.229 lb = 0.522 lb/ft^2
Total steel: 2939.458 lb = 1.986 lb/ft^2
Concrete Volume: 1791.204 ft^3 = 1.210 ft^3/ft^2

==

[3] COLUMN AXIAL FORCES AND MOMENTS

==

Units: P (kip), M (k-ft)

Supp	Case/Patt	P (axial)	Mb[top]	Ma[bottom]
1	SELF	26.21	-32.76	-34.39
	Dead	1.43	-2.04	-2.14
	Live-1	14.31	-20.40	-21.41
	Live-2	-0.07	3.05	3.21
	Live-3	10.80	-18.35	-19.26
	Live-4	11.44	-17.79	-18.68
2	SELF	50.65	5.41	5.68
	Dead	3.07	0.32	0.33
	Live-1	30.70	3.20	3.36
	Live-2	11.90	-13.76	-14.44
	Live-3	11.12	16.16	16.96
	Live-4	23.88	0.60	0.63
3	SELF	47.74	-0.97	-1.01
	Dead	2.90	-0.06	-0.06
	Live-1	29.04	-0.65	-0.69
	Live-2	10.83	15.41	16.18
	Live-3	10.96	-15.90	-16.70
	Live-4	23.60	0.01	0.01
4	SELF	47.74	0.97	1.01
	Dead	2.90	0.06	0.06
	Live-1	28.74	1.11	1.17
	Live-2	10.83	-15.41	-16.18
	Live-3	10.73	16.25	17.06
	Live-4	23.60	-0.01	-0.01
5	SELF	50.64	-5.41	-5.68
	Dead	3.07	-0.32	-0.33
	Live-1	33.97	-6.71	-7.04
	Live-2	11.90	13.76	14.44
	Live-3	13.57	-18.79	-19.72
	Live-4	26.33	-3.24	-3.40
6	SELF	26.21	32.76	34.39
	Dead	1.43	2.04	2.14
	Live-1	15.02	22.62	23.75

	Live-2	-0.07	-3.05	-3.21
	Live-3	11.33	20.02	21.02
	Live-4	11.97	19.46	20.43
Sum	SELF	249.18	-0.00	-0.00
	Dead	14.80	-0.00	-0.00
	Live-1	151.78	-0.82	-0.86
	Live-2	45.32	-0.00	-0.00
	Live-3	68.51	-0.62	-0.65

Calculation Results - Graphical Output

The following diagrams illustrate the graphical output for Example 2.

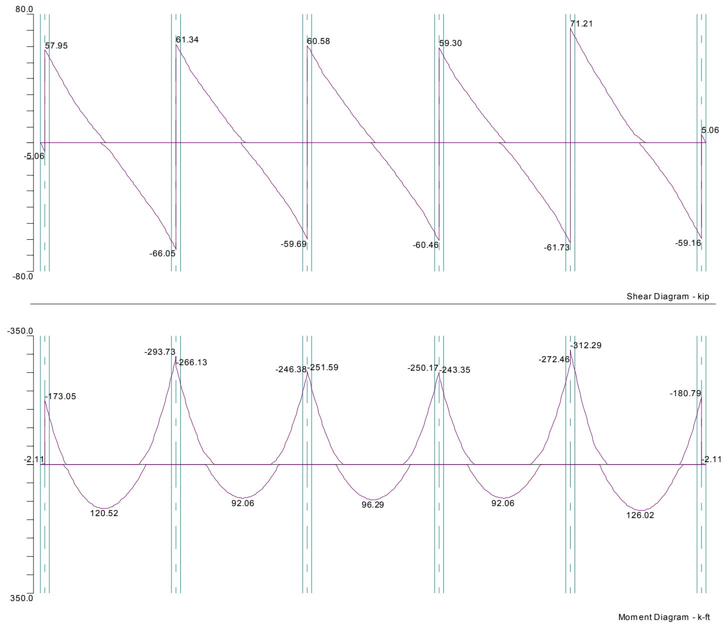


Figure 6-8 Shear and Moment Diagram

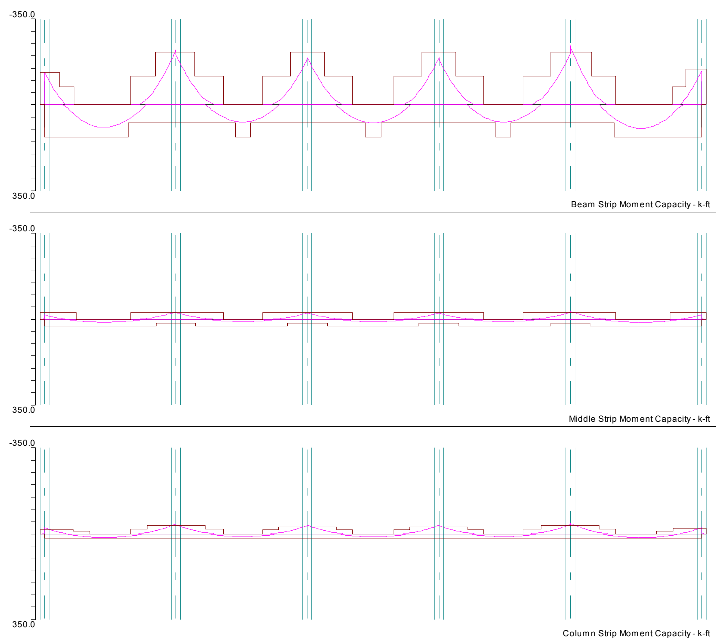


Figure 6-9 Moment Capacity Diagram

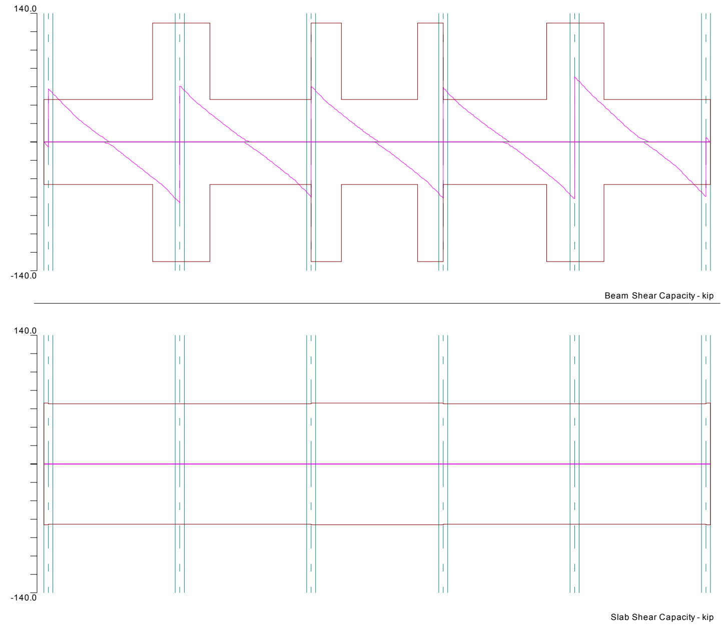


Figure 6-10 Shear Capacity Diagram

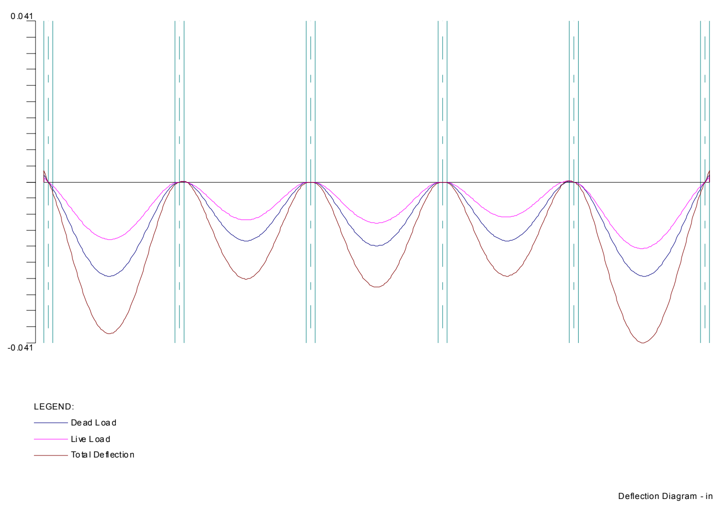


Figure 6-11 Deflection Diagram

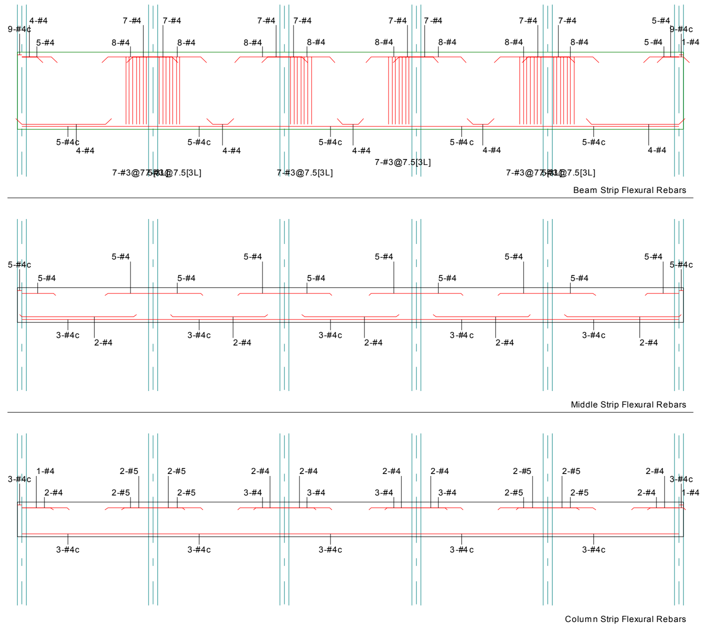


Figure 6-12 Reinforcement Diagram

Appendix

Appendix A: Conversion Factors

Conversion Factors - U.S. to SI

To convert from	To	Multiply by
in	m (1000 mm)	0.02540
ft	m	0.304800
lb	N (0.001 kN)	4.448222
Kips (1000 lbs)	kN	4.448222
plf (lb/ft)	N/m	14.59390
psi (lb/in.2)	kPa	6.89757
ksi (kips/in.2)	MPa	6.894757
psf (lb/ft2)	N/m2 (Pa)	47.88026
pcf (lb/ft3)	kg/m3	16.018460
ft-kips	kN□m	1.355818

Conversion Factors - SI to U.S.

To convert from	To	Multiply by
m (1000 mm)	In	39.37008
m	ft	3.28084
N (0.001 kN)	lb	0.224809
kN	Kips (1000 lbs)	0.224809
kN/m	plf (lb/ft)	68.52601
MPa	psi (lb/in2)	145.0377
MPa	ksi (kips/in2)	0.145038
kN/m2 (kPa)	psf (lb/ft2)	20.88555
kg/m3	pcf (lb/ft3)	0.062428
kN□m	ft-kips	0.737562

Appendix B: Bar Sizes

ASTM English Bar Sizes - English Units

Size	Diameter (in.)	Area (in.2)	Weight (lb/ft)
#3	0.375	0.11	0.376
#4	0.500	0.20	0.668
#5	0.625	0.31	1.043
#6	0.750	0.44	1.502
#7	0.875	0.60	2.044
#8	1.000	0.79	2.670
#9	1.128	1.00	3.400
#10	1.270	1.27	4.303
#11	1.410	1.56	5.313
#14	1.693	2.25	7.650
#18	2.257	4.00	13.600

ASTM English Bar Sizes - Metric Units

Size	Diameter (in.)	Area (mm2)	Weight (kg/m)
#3	9.53	71.0	0.560
#4	12.70	129.0	0.994
#5	15.88	200.0	1.552
#6	19.05	283.9	2.235
#7	22.23	387.1	3.042
#8	25.40	509.7	3.973
#9	28.65	645.2	5.060
#10	32.26	819.4	6.404
#11	35.81	1,006.5	7.907
#14	43.00	1,451.6	11.385
#18	57.33	2,580.6	20.240

ASTM Metric Bar Sizes - English Units

Size	Diameter (in.)	Area (in.2)	Weight (lb/ft)
#10	0.445	0.16	0.053
#15	0.630	0.31	0.126
#20	0.768	0.47	0.158
#25	0.992	0.78	0.264
#30	1.177	1.09	0.369
#35	1.406	1.55	0.527
#45	1.720	2.33	0.791
#55	2.220	3.88	1.319

ASTM Metric Bar Sizes - Metric Units

Size	Diameter (mm)	Area (mm2)	Weight (kg/m)
#10	11.30	100.0	0.785
#15	16.00	200.0	1.870
#20	19.50	300.0	2.356
#25	25.20	500.0	3.925
#30	29.90	700.0	5.495
#35	35.70	1,000.0	7.850
#45	43.70	1,500.0	11.775
#55	56.40	2,500.0	19.625

European Bar Sizes - English Units

Size	Diameter (in.)	Area (in.2)	Weight (lb/ft)
#6	0.236	0.04	0.015
#8	0.315	0.08	0.027
#10	0.394	0.12	0.041
#12	0.472	0.18	0.060
#14	0.551	0.24	0.081
#16	0.630	0.31	0.106
#20	0.787	0.49	0.166
#25	0.984	0.76	0.259
#28	1.102	0.95	0.325
#32	1.260	1.24	0.424
#40	1.575	1.95	0.663

European Bar Sizes - Metric Units

Size	Diameter (mm)	Area (mm2)	Weight (kg/m)
#6	6.00	28.3	0.222
#8	8.00	50.3	0.395
#10	10.00	78.5	0.617
#12	12.00	113.0	0.888
#14	14.00	154.0	1.208
#16	16.00	201.0	1.578
#20	20.00	314.0	2.466
#25	25.00	491.0	3.853
#28	28.00	616.0	4.836
#32	32.00	801.0	6.313
#40	40.00	1,256.0	9.865

Appendix C: References

References

1. Kripanarayanan, K. M., Analysis and Design of Slab Systems, Computer Program User's Manual, Portland Cement Association, SR184D, 1979.

2. Lee, B. H., Analysis and Design of Flat Plates, Flat Slabs, Waffle Slabs, and Continuous Frames, Portland Cement Association, 1972 (discontinued).

3. CRSI Handbook Based Upon the 1977 ACI Building Code and the 1980 Supplement, Concrete Reinforcing Steel Institute, Schaumburg, Illinois, 60195, 1982.

4. Sozen, M. A., and Siess, C. P., Reinforced Concrete Floor Slabs – Design Methods: Their Evolution and Comparison, Journal of the American Concrete Institute, Proceedings, V. 60, No. 8, August 1963, pp. 999-1028.

5. Hatcher, D. S., Sozen, M. A., and Siess, C. P., Test of a Reinforced Concrete Flat Plate, Proceedings of the American Society of Civil Engineers, V. 91, No. ST5, October 1965, pp. 205-231.

6. Guralnick, S. A., and LaFraugh, R. W., Laboratory Study of a Forty-Five-Foot Square Flat Plate Structure, Journal of the American Concrete Institute, Proceedings, V. 60, No. 9, September 1963, pp. 1107-1185.

7. Hatcher, D. S., Sozen, M. A., and Siess, C. P., Test of a Reinforced Concrete Flat Slab, Proceedings of the American Society of Civil Engineers, V. 95, No. ST6, June 1969, pp. 1051-1072.

8. Jirsa, J. O., Sozen, M. A., and Siess, C. P., Test of a Flat Slab Reinforced with Welded Wire Fabric, Proceedings of the American Society of Civil Engineers, V. 92, No. ST3, June 1966, pp. 199-224.

9. Gamble, W. L., Sozen, M. A., and Siess, C. P., Test of a Two-Way Reinforced Floor Slab, Proceedings of the American Society of Civil Engineers, V. 95, No. ST6, June, 1969, pp. 1073-1096.

10. Vanderbilt, M. D., Sozen, M. A., and Siess, C. P., Test of a Modified Reinforced Concrete Two-Way Slab, Proceedings of the American Society of Civil Engineers, V. 95, No. ST6, June, 1969, pp. 1097-1116.

11. Shewmaker, R. E., Xanthakis, M., and Sozen, M. A., Very Small Scale Reinforced Concrete Multi-Panel Flat Slabs, Civil Engineering Studies, Structural Research Series No. 265, University of Illinois, June 1963, 171 pp.

12. Xanthakis, M., and Sozen, M. A., An Experimental Study of Limit Design in Reinforced Concrete Flat Slabs, Civil Engineering Studies, Structural Research Series No. 277, University of Illinois, December 1963, 159 pp.

13. Nichols, J. R., Statistical Limitations Upon the Steel Requirement in Reinforced Concrete Flat Slab Floors, Transactions of the American Society of Civil Engineers, V. 77, 1914, pp. 1670-1726.

14. Gamble, W. L., Sozen, M. A., and Siess, C. P., Measured and Theoretical Bending Moments in Reinforced Concrete Floor Slabs, Civil Engineering Studies, Structural Research Series No. 246, University of Illinois, June 1962, 322 pp.

15. Jirsa, J. O., Sozen, M. A., and Siess, C. P., Pattern Loadings on Reinforced Concrete Floor Slabs, Proceedings of the American Society of Civil Engineers, V. 95, No. ST6, June, 1969, pp. 1117-1137.

16. Corley, W. G., Sozen, M. A., and Siess, C. P., The Equivalent Frame Analysis for Reinforced Concrete Slabs, Civil Engineering Studies, Structural Research Series No. 218, University of Illinois, June, 1961, 166 pp.

17. Jirsa, J. O., Sozen, M. A., and Siess, C. P., The Effects of Pattern Loadings on Reinforced Concrete Floor Slabs, Civil Engineering Studies Structural Research Series No. 269, University of Illinois, July 1963, 145 pp.

18. Corley, W. G., and Jirsa, J. O., Equivalent Frame Analysis for Slab Design, Journal of the American Concrete Institute, Proceedings, V. 67, No. 11, November 1970, pp. 875-884.

19. ETABS84, Computers and Structures, Inc., Berkeley, California, 1984.

20. ICES STRUDL II, Engineering User's Manual, Vols. 1 and 2, Department of Civil Engineering, Massachusetts Institute of Technology, R68-91/92, Cambridge, Massachusetts, 1968.

21. Hawkins, N. M., Shear Strength of Slabs with Moments Transferred to Columns, Special Publication SP42-35, American Concrete Institute, Detroit, Michigan, 1974, pp. 817-846.

22. ACI Committee 435, Subcommittee 7, Deflections of Continuous Concrete Beams, Journal of the American Concrete Institute, Proceedings V. 70, No. 12, December 1973, pp. 781-787.

23. Nilson, A. H., and Walters, D. B., Jr., Deflection of Two-Way Floor Systems by the Equivalent Frame Method, Journal of the American Concrete Institute, Proceedings, V. 72, No. 5, May, 1975, pp. 210-218.

24. ACI Committee 435, State-of-the-Art Report, Deflection of Two-Way Floor Systems, Special Publication SP43-3, American Concrete Institute, Detroit, Michigan, 1974.

25. Kripanarayanan, K. M., and Branson, D. E., Short-Time Deflections of Flat Plates, Flat Slabs and Two-Way Slabs, Journal of the American Concrete Institute, Proceedings, V. 73, No. 12, December 1976, pp. 686-690.

26. Branson, D. E., Instantaneous and Time-Dependent Deflections of Simple and Continuous Reinforced Concrete Beams, HPR Report No. 7, Pt. I, Alabama Highway Department in Cooperation with U.S. Department of Commerce, Bureau of Public Roads, August 1965.

27. Blakey, F. A., Deformations of an Experimental Lightweight Flat Plate Structure, Commonwealth Scientific and Industrial Research Organization (CSIRO), DBR Reprint No. 185, Civil Engineering Transactions, Institute of Engineers, V. CE 3, No. 1, Australia, March 1961.

28. Taylor, P. J., Long-Term Deflection Calculation Methods for Flat Plates, Constructional Review, V. 43, No. 2, Sydney, Australia, May 1970, pp. 68-74.

29. Heiman, J. L., A Comparison of Measured and Calculated Deflections of Flexural Members in Four Reinforced Concrete Buildings, Special Publication JSP43-21, American Concrete Institute, Detroit, Michigan, 1974.

30. Rangan, B. V., Prediction of Long-Term Deflections of Flat Plates and Slabs, Journal of the American Concrete Institute, Proceedings, V. 73, No. 4, April 1976, pp. 223-226.

31. Notes on ACI318-83 – Building Code Requirements for Reinforced Concrete with Design Applications, Portland Cement Association, 1984.

32. Elgabry, A. and Ghali, A., Proposed Revisions to Building Code Requirements for Reinforced Concrete (ACI 318-83) (Revised 1986), ACI Structural Journal, V. 86 No. 3, May-June 1989, pp. 327-328.

33. Notes on ACI318-95 – Building Code Requirements for Reinforced Concrete with Design Applications, Portland Cement Association PCA, 1996.

34. Manual of Standard Practice, Concrete Reinforcing Steel Institute CRSI, March 2001.

35. Reinforced Concrete, Mechanics & Design, 3rd ed. James McGregor, Prentice Hall, 1997.

36. Nilson, A. and Winter, G., Design of Concrete Structures, McGraw-Hill, New York, 1997.

37. Wang, C. K. and Salmon, C. G., "Reinforced Concrete Design" 6th Ed., Addison-Wesley, 1998.

38. A23.3-94 Design of Concrete Structures, Structures Design, Canadian Standards Association CSA, December 1994.

39. Concrete Design Handbook, Cement Association of Canada CAC, 2nd Ed., 2001.

40. Building Code Requirements for Structural Concrete (318-99) and Commentary (318R-99), American Concrete Institute ACI, Detroit, Michigan, June 1999.

41. Building Code Requirements for Structural Concrete (318-02) and Commentary (318R-02), American Concrete Institute ACI, Detroit, Michigan, January 2002.

42. James R. Cagley, Changing from ACI 318-99 to ACI 318-02 What's New? Concrete International, June 2001.

Index

O

P

R

S

T

V