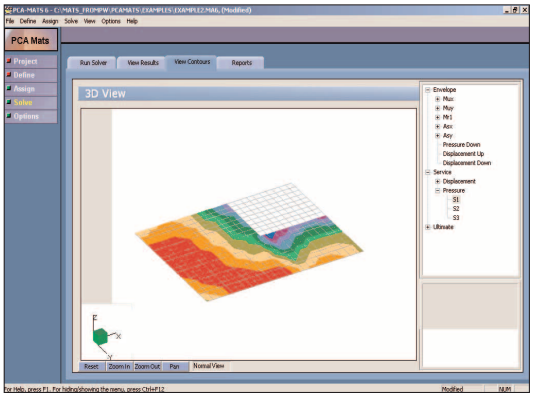
pcaMats v6.0

For the analysis of foundation mats, combined footings, or slabs on grade



pcaMats analyzes foundation mats, combined footings, or slabs on grade. The slab is modeled as an assemblage of rectangular finite elements. The boundary conditions may be the underlying soil, nodal springs, piles, or translational and rotational nodal restraints. The model is analyzed under static loads that may consist of uniform (surface) and concentrated loads. The resulting deflections, soil pressure (or spring reactions), and bending moments are output. In addition, the program computes the required area of reinforcing steel in the slab.

pcaMats uses the plate-bending theory and the Finite Element Method (FEM) to model the behavior of the mat or slab. The soil supporting the slab is assumed to behave as a set of one-way compression-only springs (Winkler foundation). If, during the analysis, a loading or the mat shape causes any uplift creating a spring in tension, the spring is automatically removed. The mat is re-analyzed without that or any other tension spring. The program automatically iterates until all springs are in compression and the foundation stabilizes.

A new feature in the program allows the user to define rectangular or circular column sections and assign the columns to different locations to calculate the punching shear.

System Requirements

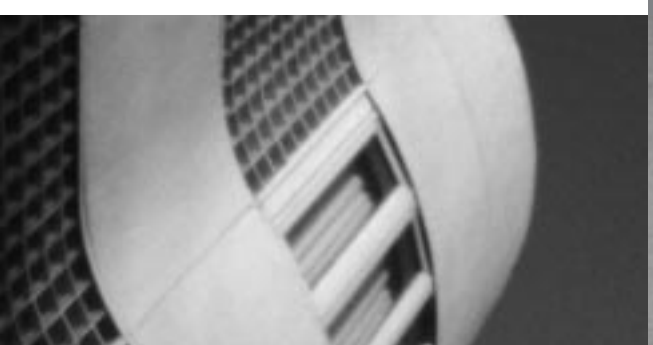
- Microsoft Windows 98 or higher

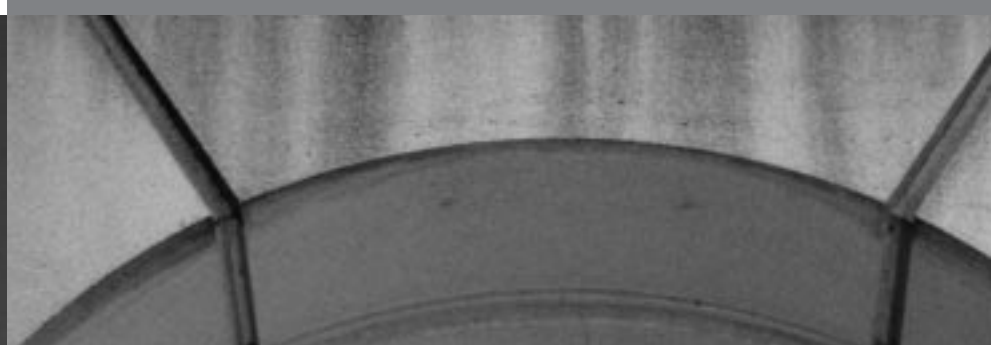
pcaStructurePoint

Work quickly. Work simply. Work accurately.

PCA
Portland Cement Association

www.pcastructurepoint.com





Input

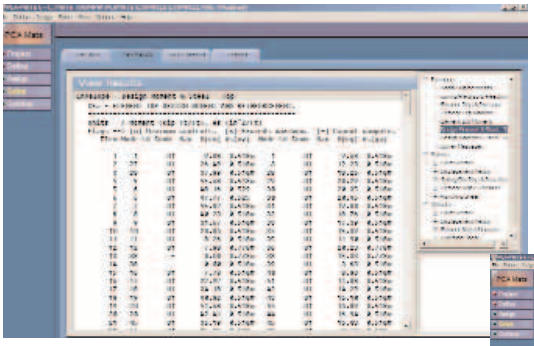
In pcaMats, a rectangular grid layout that defines the slab mesh is easily input. The slab geometry is then created by graphically applying element thickness to the grid system. The program automatically numbers all nodes and elements.

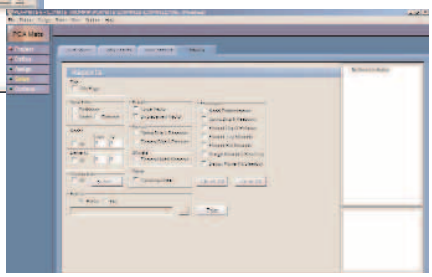
pcaMats employs a graphical interface that allows the user to graphically define the slab geometry, apply the material properties, boundary conditions, and the loads. The basic problem definitions are first input using dialog boxes that are clearly labeled. These definitions are then applied to the slab.

All input parameters are entered in databases using dialog boxes. A basic grid system is defined by entering the grid line coordinates with the possibility of generating equally spaced grid-lines. The finite element mesh is generated by graphically applying element thickness to the grid system. All assignments, material properties (concrete, soil, and reinforcing steel), nodal springs, nodal restraints, slaved nodes, nodal loads, and element loads are also graphically applied to the model using the mouse.

Technical Features

- Four-noded prismatic thin plate element with three degrees of freedom per node
- Material properties (concrete and reinforcing steel) may vary from element to element
- Soil may be applied uniformly over elements or concentrated at nodes (nodal spring supports)
- Nodes may be restrained for vertical displacement and/or rotation about X and/or Y
- Nodes may be slaved to share the same displacement and/or rotation
- Loads may be uniform (vertical force per unit area) or concentrated (Pz, Mx, and My)
- Six load cases and practically unlimited load combinations
- Load combinations are categorized into service (serviceability) and ultimate (design) levels
- The self weight of the slab is automatically computed and may optionally be excluded in the analysis
- Result envelopes (maximum and minimum values) for deflections, pressures, and moments
- Design moments include torsional moments contribution
- Punching shear calculations for rectangular and circular columns





Graphical Interface Features

- Fast graphical interface that displays the modeled mesh at all times for verification
- Graphical image displaying node and element numbers, grid lines, and slab boundaries
- Ability to zoom and translate (pan) the graphical image
- Isometric (3D) view of the modeled slab with ability to rotate using the mouse
- User-controlled screen color settings
- Contour plots

Output

- Ability to view results in text and graphical format prior to printing
- Contour plots for deflections, pressures, and moments
- Selective printing allowing printing of results for all or user-selected nodes, members, and load combinations
- Echo of input data
- Generated finite element mesh data
- Nodal displacements and rotations
- Nodal spring displacement and reaction
- Soil displacement and pressure
- Element nodal moments including rectangular, torsional, and principal moments
- Nodal displacement envelopes for all service level load combinations
- Spring displacement and reaction envelopes
- Soil displacement and pressure envelopes
- Element moment envelopes
- Design moment envelopes with required area of reinforcing steel

Program Capacity

- Up to 255 entries in each database (concrete, steel, soil, springs, etc.)
- 255 surface loads and 255 nodal loads per load case
- 255 service and 255 ultimate load combinations
- Up to 10,000 nodes

General Features

- Online help
- English or metric unit system
- Ability to save settings and defaults for future input sessions
- Data checked as they are input
- Automatic internal node and setting numbering

www.pcastructurepoint.com

Visit our Web site for updates, technical papers, new products, technical support, newsletters, and demos.