BE4E – PLPAK

Towards more realistic structural modeling

PLPAK introduction & Main Package

Table of content for lecture 1

- Introduction.
  - What is PLPAK?
  - Boundary elements & Finite elements.
  - Why are we using boundary element via PLPAK?
- PLPAK Package (Basic).
  - The pre-processor file (PLGen).
    - How can we build a model?
  - Check model (PLView).
    - the model in PLView.
  - Run model (PLCoreMan).
    - Manager of PLPAK.

What is PLPAK?

• The PLPAK is special purpose software package for structural analysis of building slabs and foundations. The PLPAK uses the boundary element method as numerical method.

• The PLPAK solves single floor at the time; each floor consists of single slab with several openings.

• The PLPAK deals with the real geometry of structural element so we can reach for more realistic simulation for the structure.

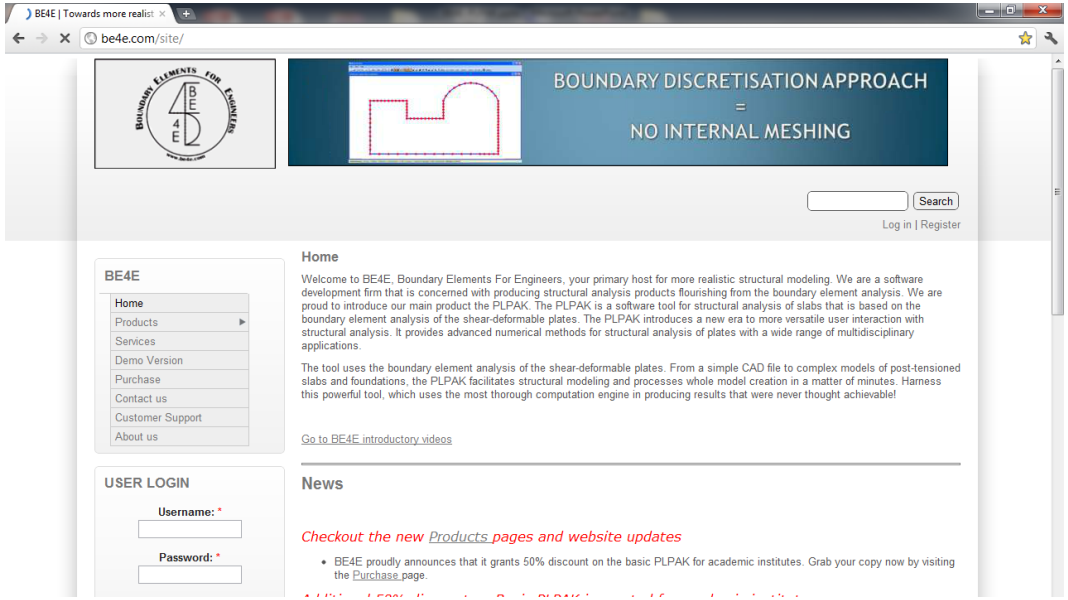
• The PLPAK is very easy to learn and doesn't require any previous knowledge of boundary elements .

What is BE4E?

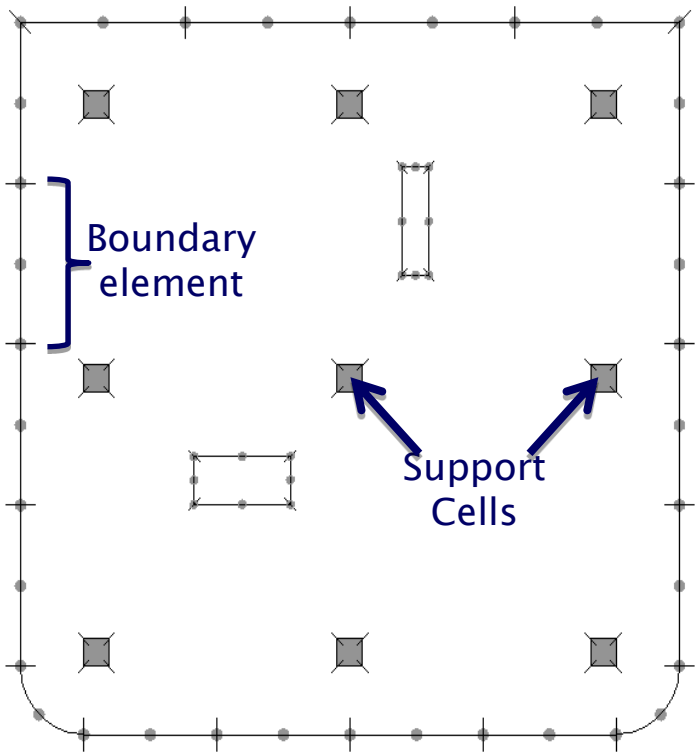
• BE4E (Boundary Element for Engineers) is a group professors & engineers interested in boundary element analysis.

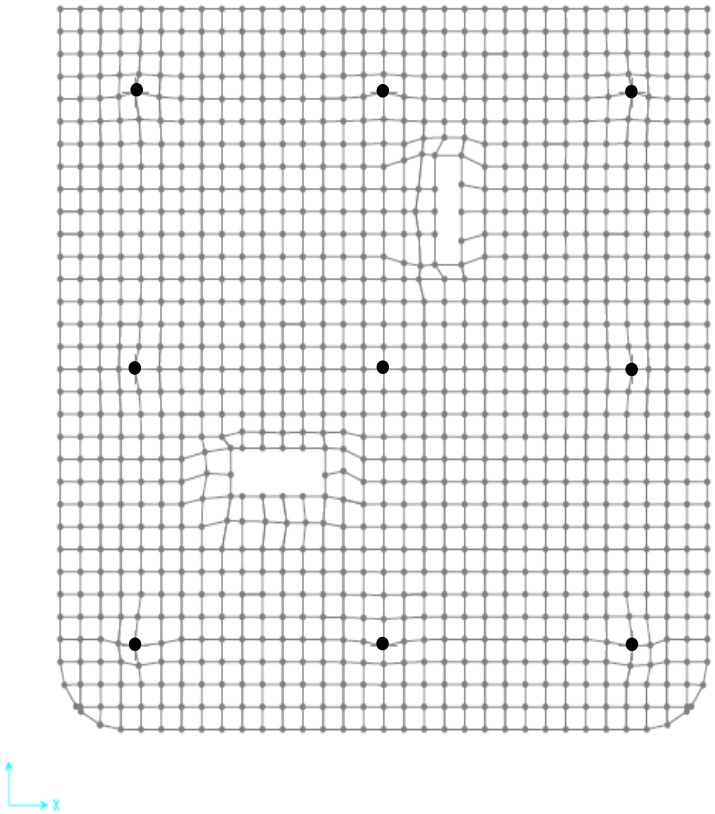
• Our vision is to see structural analysis techniques become more realistic and easier for practicing engineers.

• Our mission is to bring new numerical approaches to model building slabs and foundation plates, and to continue updating these approaches with our developed new R&D results.



Boundary element & Finite element





BEM model

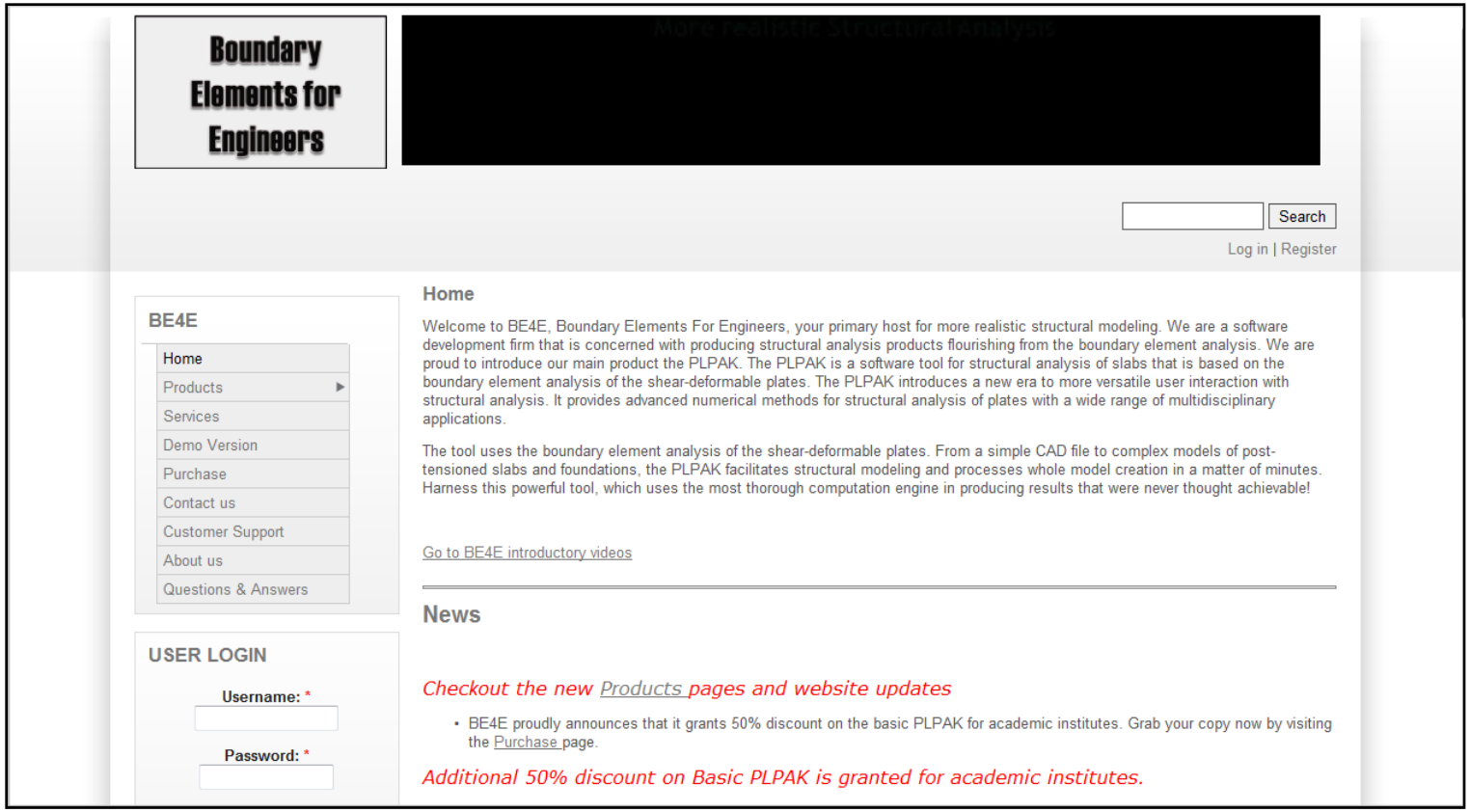
FEM model

Why are we going to use BE via PLPAK?

We are using PLPAK for the following reasons:

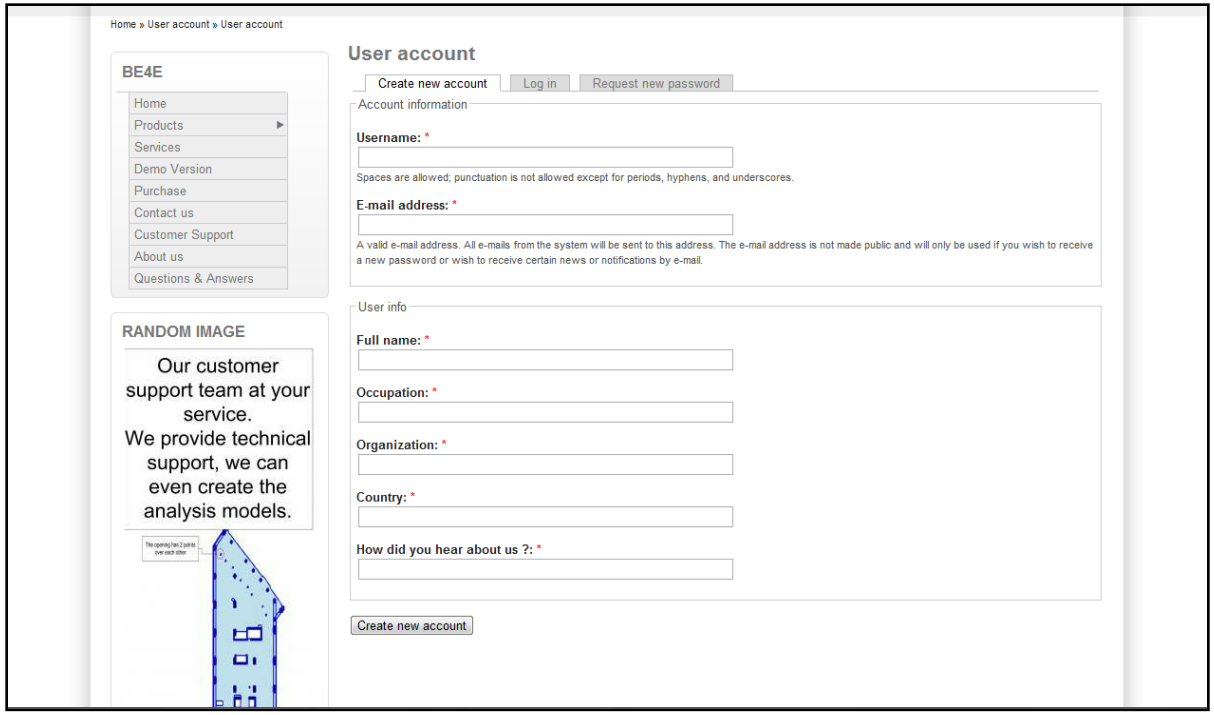
- More accurate as it deals with real dimensions of the structural elements (No peak values).
- Real time post-processing.
- Easy input in forms of:
  - Multiple exporting from DXF to the virtual model in the PLGen, makes life easy for engineers to account for any modification in design. This is done without re-meshing or re-building the numerical model.
  - Multiple level of "undo" in the PLPAK preprocessor.
- Capable of modeling fine details such as small duct openings.
- Easy drawing capabilities of the PLPAK.
- The PLPAK models beams with it's actual interaction area to the slab which make it unique in case of modeling slab with irregular beams supported not on the full beam width.
- The soil springs in soil models are considered as continuous spring patches underneath the continuum foundation plate.

BE4E WebSite



- www.be4e.com is the website that link between the PLPAK users and the customer support.
- From the website, the users can see the video tutorials for the PLPAK and can ask any questions about the boundary element or about the PLPAK Package.
- In the website, the users see the latest newsletters and the latest products and can join us on face book, Wikipedia and LinkedIn.

How can the user link with the customer support?

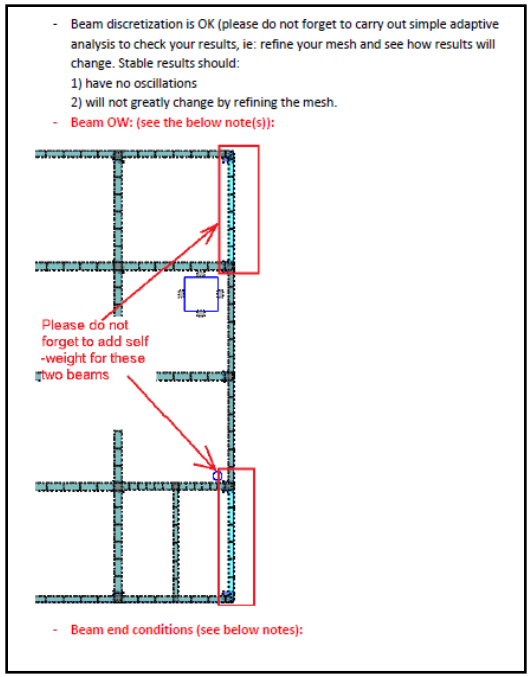


- All PLPAK users should register in the website to be able to contact with us.
- It's vey important to answer the question clearly to be able to confirm your account.

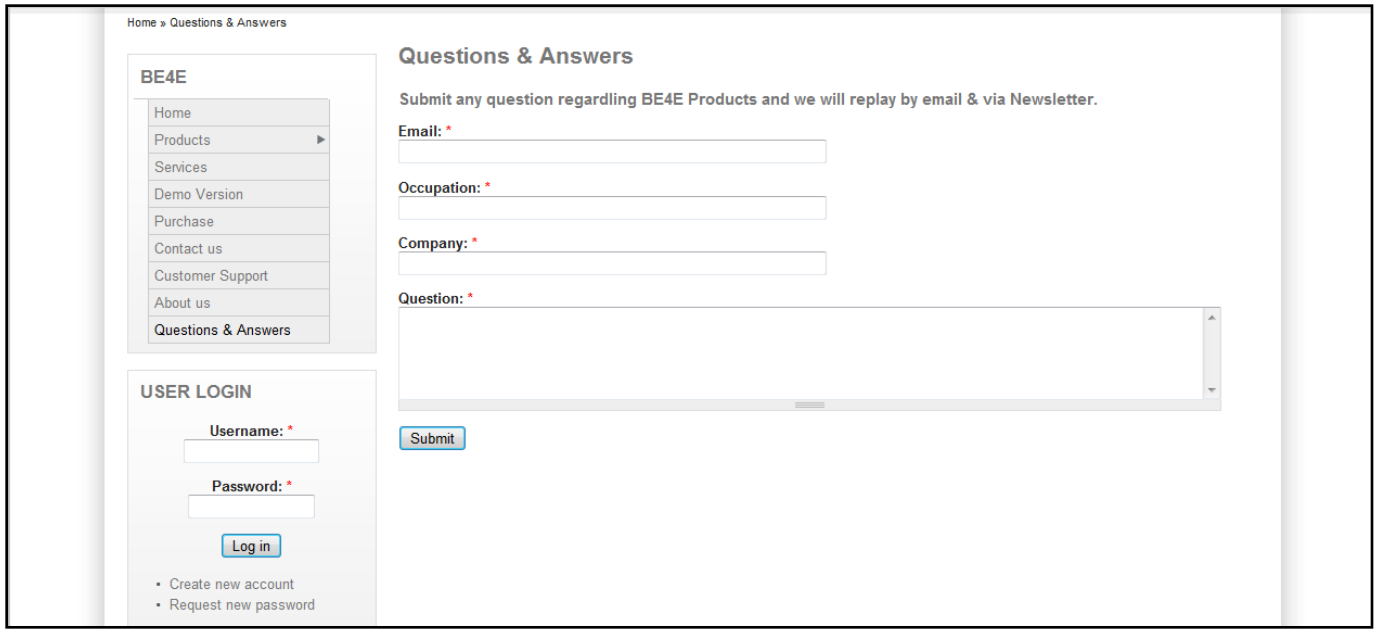
What happen if there is a problem in PLPAK?

The user should send email to support@be4e.com, this email should contain the Gen file and description of the problem and it will be better to send a print screen for the problem.

Once the user send the email, the PLPAK group will see the problem and solve it then the user will receive an email showing the solve of the problem.



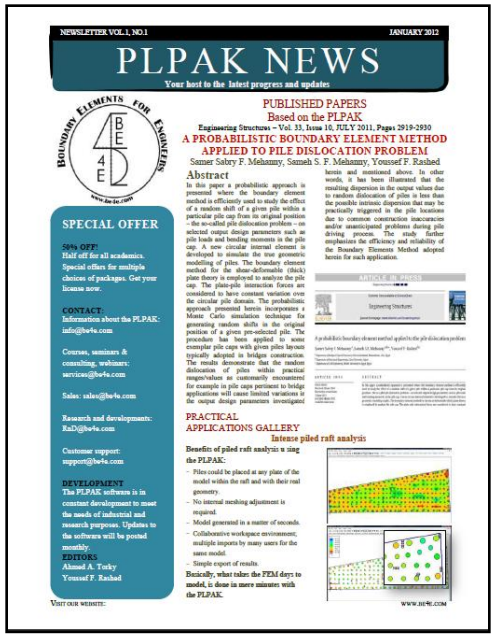
If the user have any question about the Boundary elements methodology.

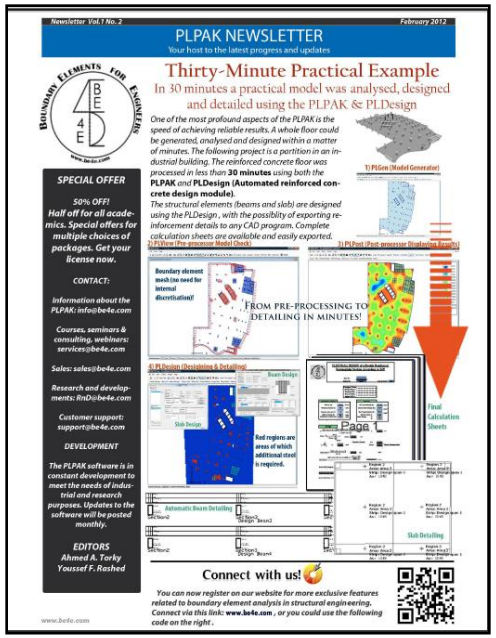


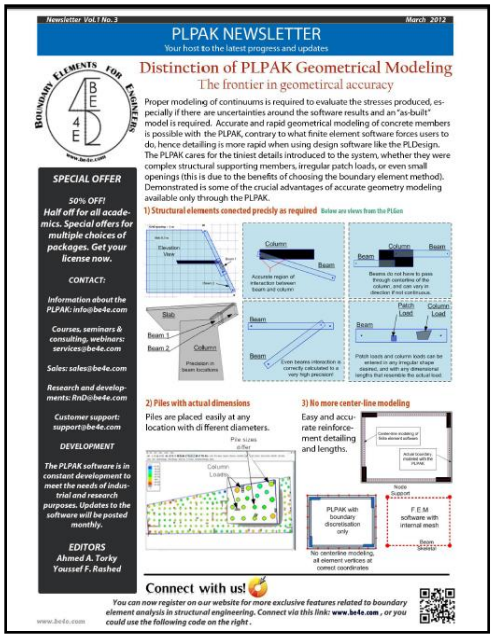
After registration the user can send any question about the methodology of the boundary elements and it will be answered and send to the user by email .

News letters

Once the registration is confirmed, the user will be able to receive every month a news letter from the PLPAK this news letter containing information about PLPAK Packages.







Course Content

• Main Package (PLPAK) (2 Lectures).

- This Package will learn us How can we make a model using PLPAK.
- Showing the stresses on different structural elements.

• Design Package (PLDesign) (2 Lectures).

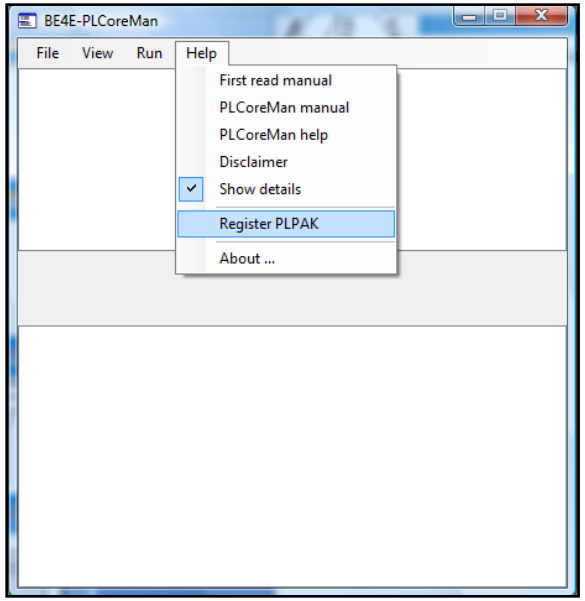
- This Package will learn us How can we design using different codes.
- Learning How can we insert/show reinforcement bars for different types of elements.

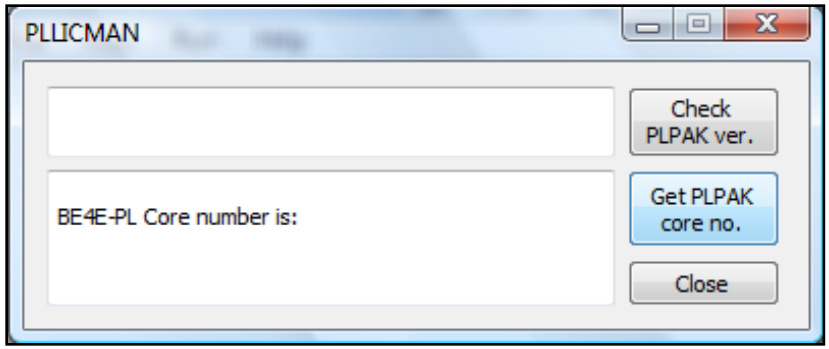
• Elastic Half Space Package (EHSPAK) (1 Lecture).

- This Package will learn us What is Elastic half space and What is the different between EHS and Winkler models.
- Learning How can we use Elastic half space model.

PLPAK Installation

After running setup of the PLPAK, go to start menu then open PLCoreMan.



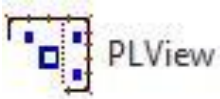


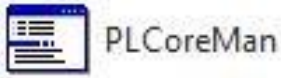
Send the core number to yrashed@be4e.com then the user will receive mail containing the PLPAK license.

Main package (PLPAK)

The PLPAK contains four different types of files:

1- Generator file (PLGen). 

2- Boundary element viewer (PLView). 

3- Manger file (PLCoreMan). 

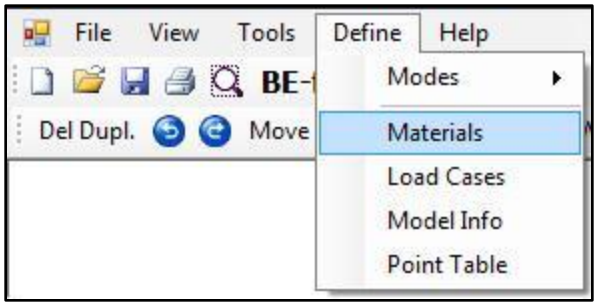
4- Post-processing file (PLPost). 

Generator file (PLGen)

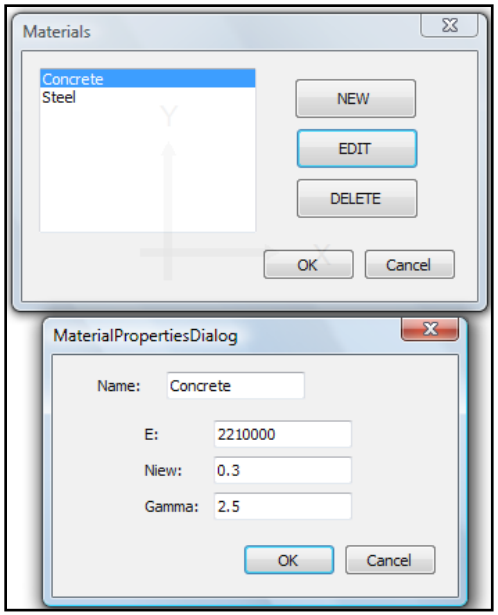
The PLGen is responsible for building the model showing every structural element this will lead us to a big question How can we Build a model?

<u>First step</u>, the user are going to adjust the model information

In the model information we will define material properties from material tab



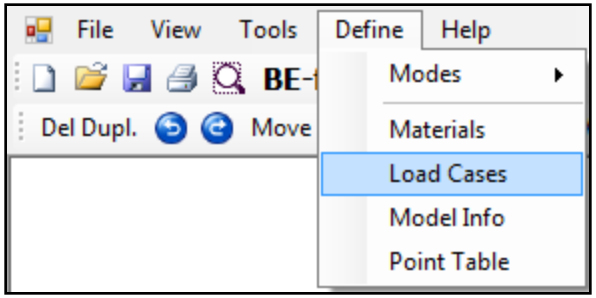
OR



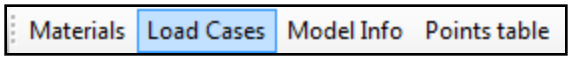
The user can define the properties of the construction material such as Young's modulus, Poisson's ratio and Gamma of the materials.

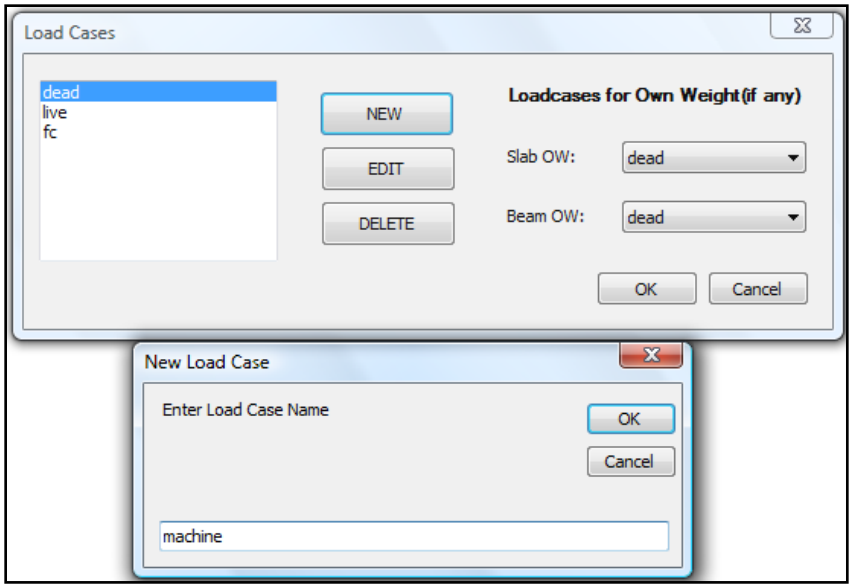
How can we build a model?

Load cases information is also a type of model information which is determined by user.



OR

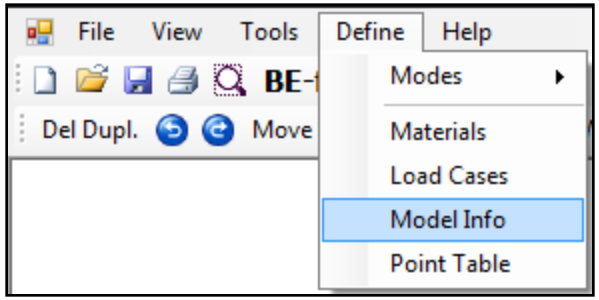




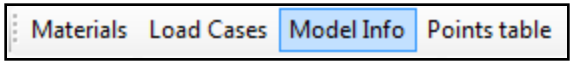
The user can define different load cases and which one can include the slab and beam own weight.

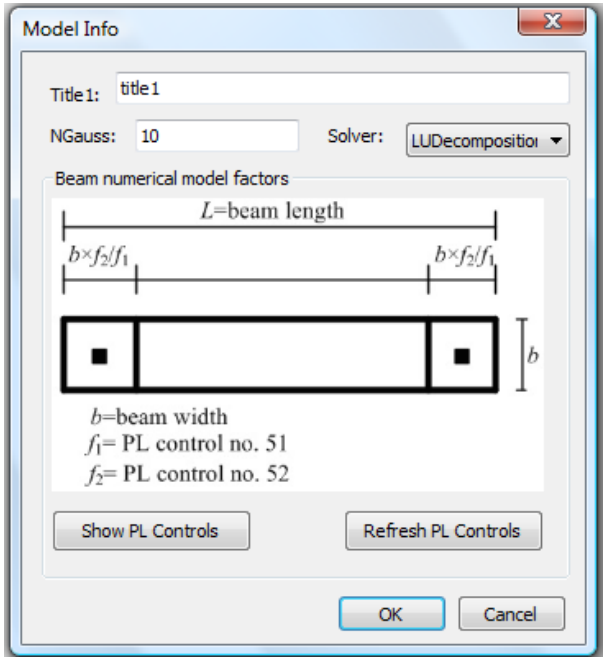
How can we build a model?

There are another information can be changed from model info tab.



OR





- The user can change number of Gauss points to decrease time consumption in modeling but the number should be <u>even number</u>.
- The user can change the type of solver either LUDecompostion or GaussElemination .
- Also the user can change the PL Controls. (take about it later).

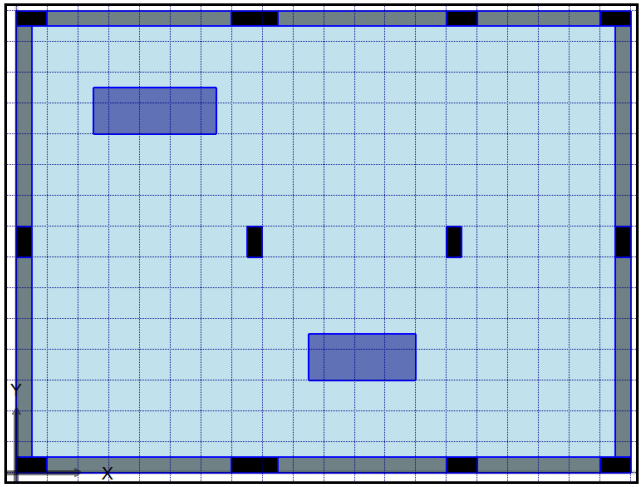
How can we build a model?

<u>Second step</u> is insert the model in the Generator file

The user can insert the model by two ways

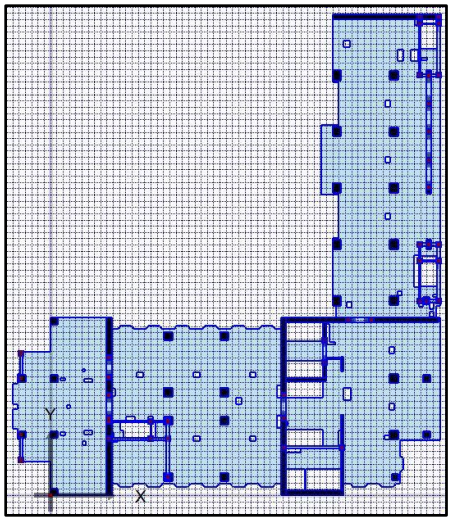
Drawing Model from PLGen

This option is useful for small models.



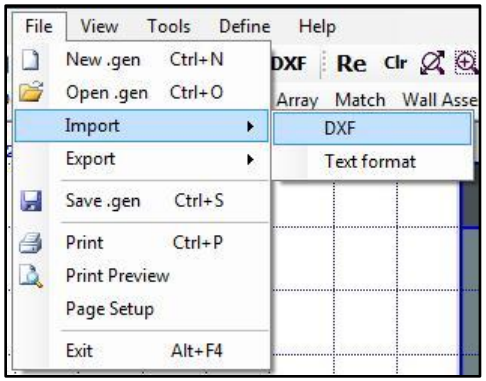
Importing (.Dxf) file (AutoCAD file)

This option is useful for large models or detailed models.



How can we build a model?

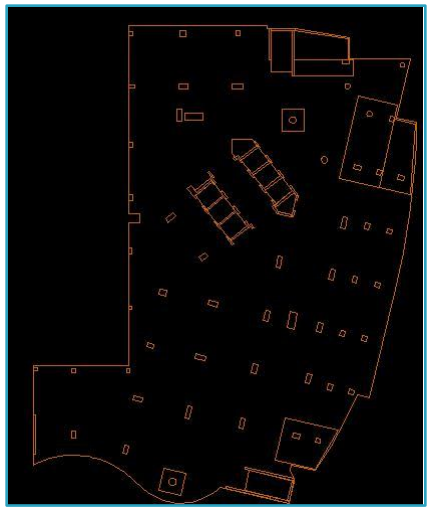
Importing (.Dxf) file is from file menu then Import Dxf.

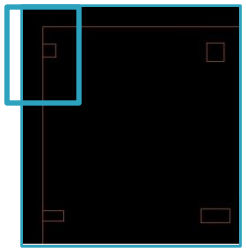


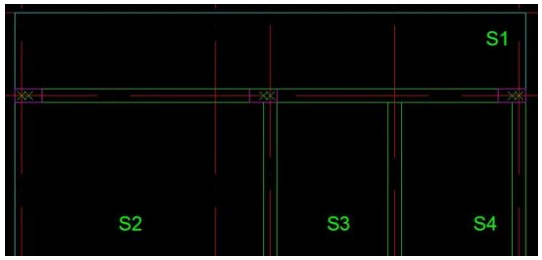
OR

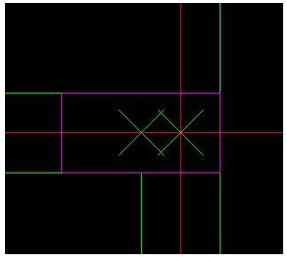
There are important notes that should be taken into consideration before Importing (.Dxf) file.

1- The slab line should be the outer line for the drawing.







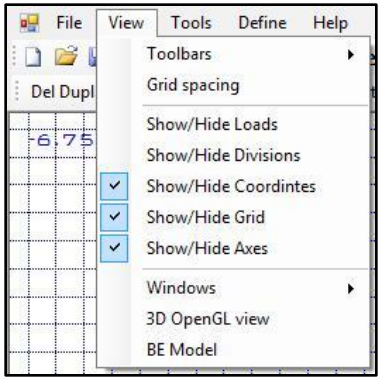


2- All structural elements should be polyline in AutoCAD.

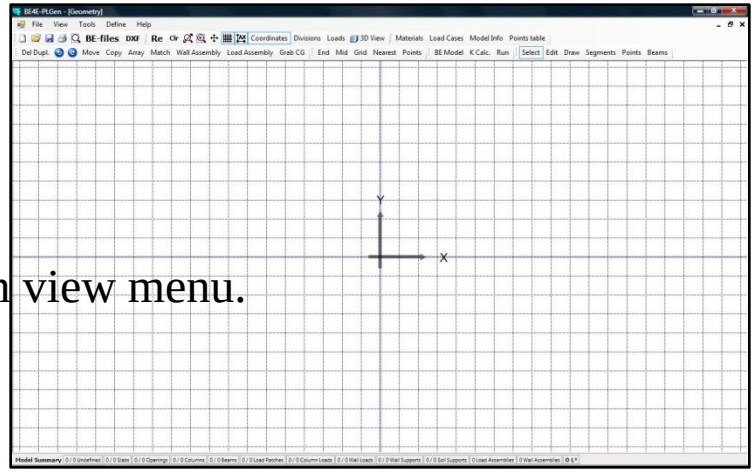
3- Beams should be inserted in AutoCAD is points at start/end of the beams.

Drawing Model from PLGen

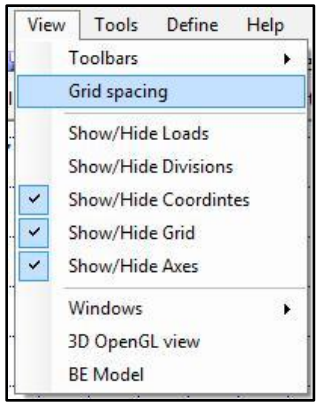
1- The user should show the Grids and show Coordinates.

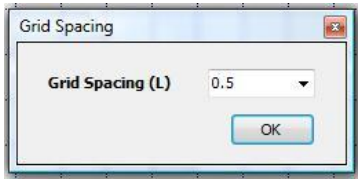


OR



The user can change the Grid spacing from view menu.



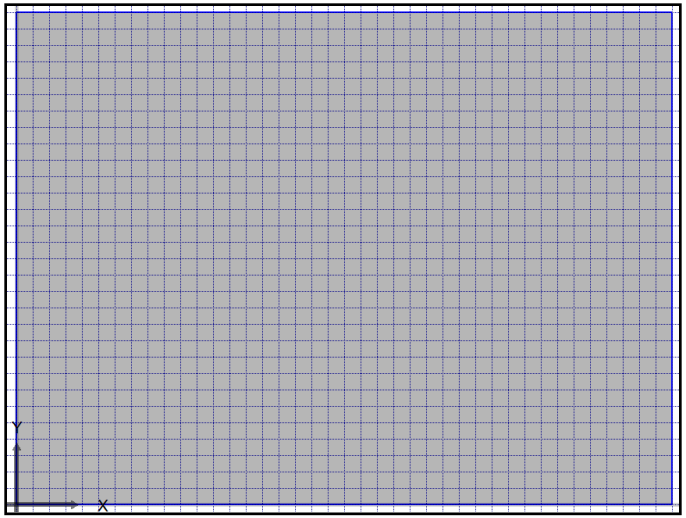


2- Select Draw tab and then Grid tab to draw the slab on the selected grids.

How can we build a model?

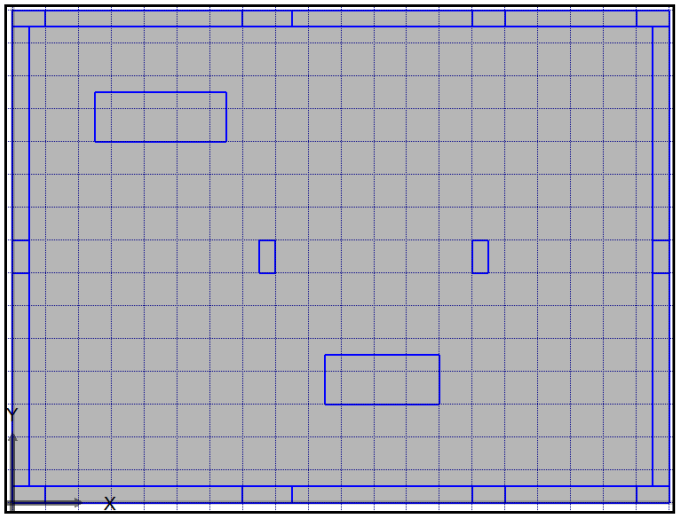
3- Drawing the elements on the grids.

Starting with the main slab by pressing left click on the four corners and closing with right click



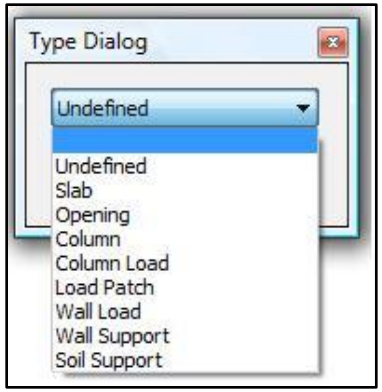
It doesn't matter if we draw clockwise or anticlockwise.

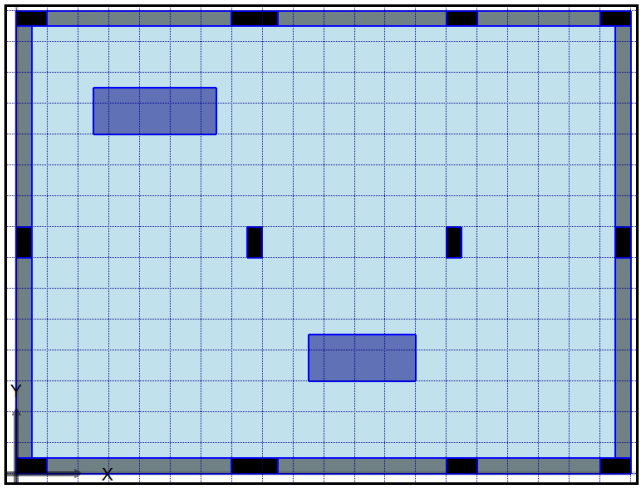
Then draw the support, load elements, and the opening but still we didn't define them.



How can we build a model?

Third step is defining different elements by choosing select tab then left click to select the element then right click to choose the type of the element.





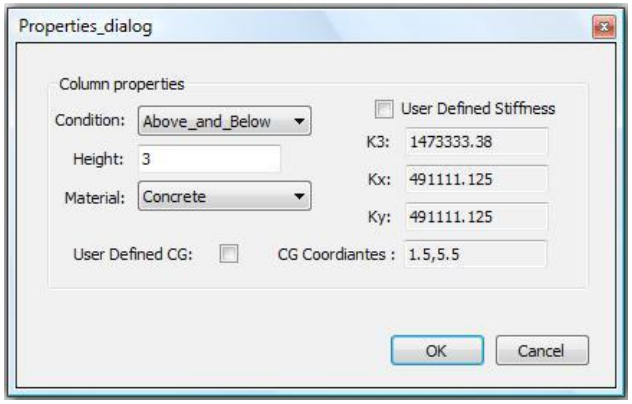
This will face us with a second question what is the different between column, wall support and soil support?

And what are the difference between column load, wall load and load patch?

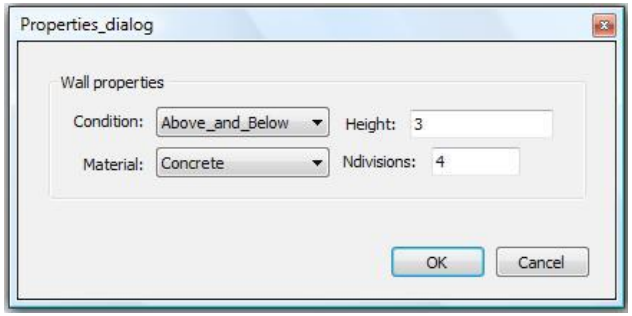
What are the difference between different types of supports?

The different between columns, wall support and soil support is the stiffness.

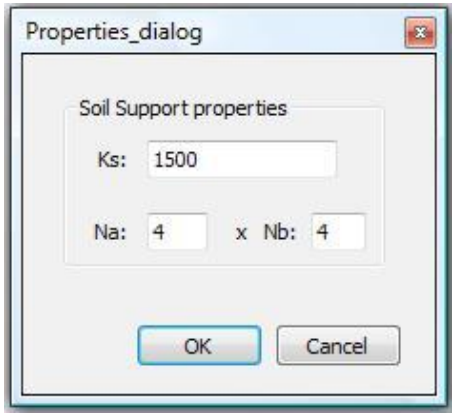
If we clicked right on a column element to show the properties, we will see that the columns contains 3 stiffness in all directions.



While the wall element is consisting of a group of columns combining the stiffness together to form the wall.



In case of soil support we will see it contains 1 stiffness in z-direction only.



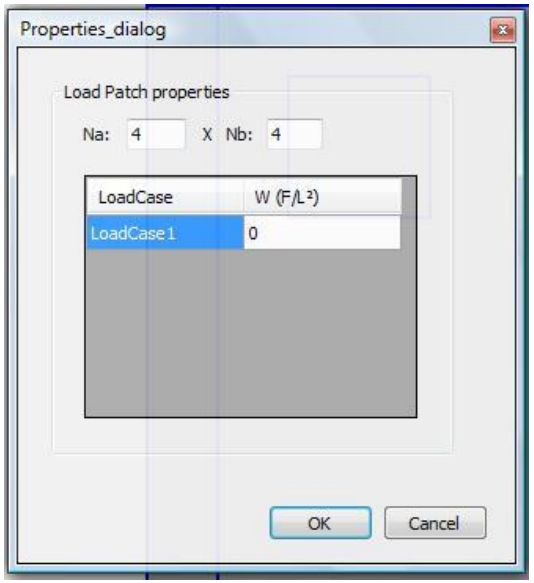
What are the difference between different types of loads?

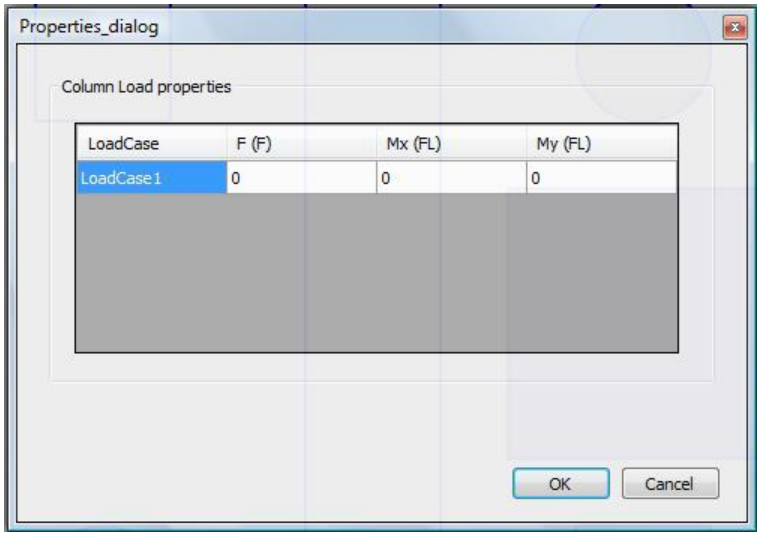
The different between Load patch, column load and wall load is the dimensions.

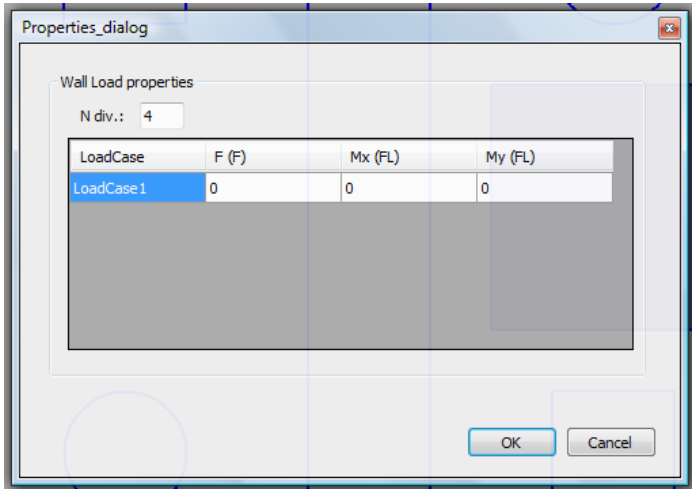
If we clicked right on load patch to show the properties, we will see that the load patch is per square length.

While the wall load is the load divided on a certain number of columns combined together.

In case of column load we will see it is load without any length dimensions.



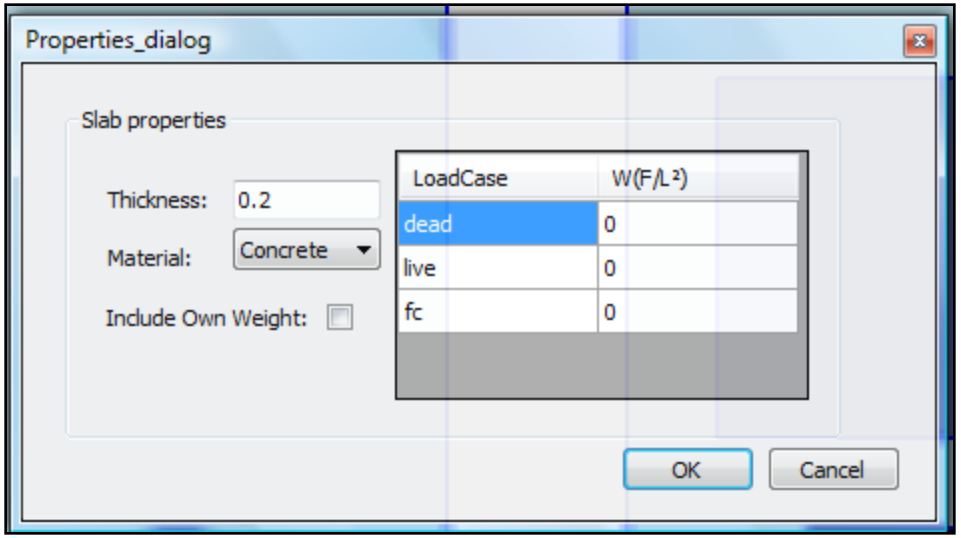




How can we build a model?

After defining support and load element, it is time do define slab thickness and define load on slab.

Press right click on the slab to show the slab properties.



From the properties dialog for slab, the user can insert slab thickness, the material of the slab, insert own weight either by check box or write the weight of slab in the Load Case (dead), insert if any distributed load on slab.

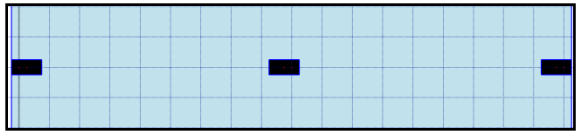
Beams

In case of solid slab or margin beams in flat slab we should know how to insert the beams in the Gen file.

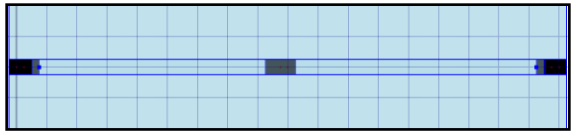
We can insert beams by two ways

Drawing from PLGen

Using the snap tools.

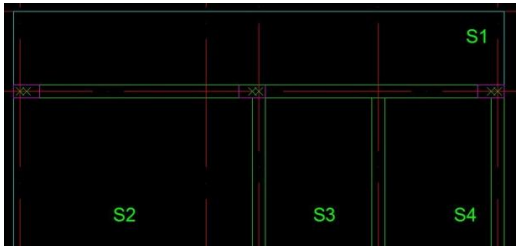


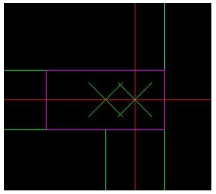
End | Mid | Grid | Nearest | Points | BE Model | K Calc. | Run | Select | Edit | Draw | Segments | Points | Beams



Importing (.Dxf) file (AutoCAD file)

Import the beam start/end as points from AutoCAD.



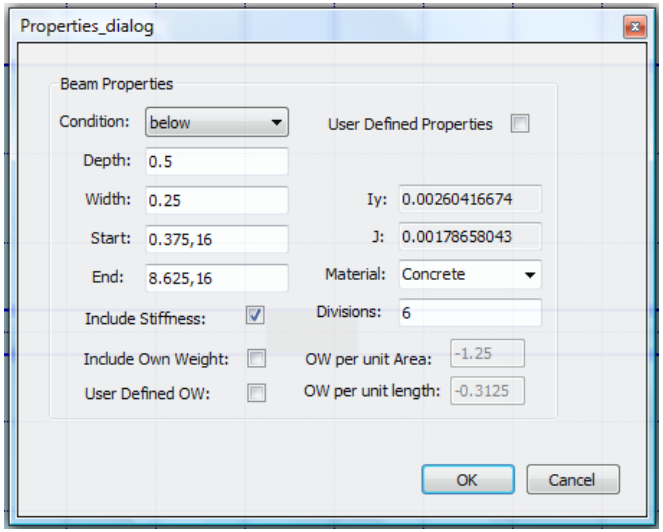


End | Mid | Grid | Nearest | Points | BE Model | K Calc. | Run | Select | Edit | Draw | Segments | Points | Beams

Beams

Once we draw all beams in the slab system we should see its properties.

For showing beam properties the user should change from beam mode to select mode, then right on the beam.



The user can change the beam condition (below or above), the beam dimensions, include stiffness (incase of loading beam), include Own Weight, number of division, user defined properties (moment of inertia & torsion constant).

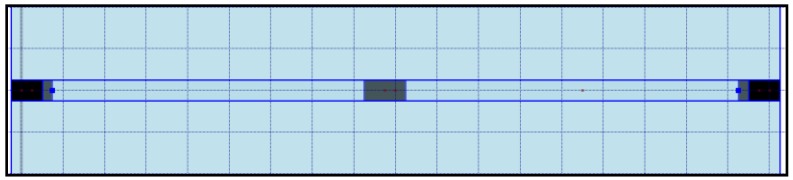
In beams there is a questions what is the different between hinged and fixed beams?

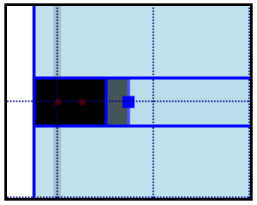
Beams

The user has an option to make the beams in his model either hinged or fixed.

In case of hinged beams

The beam inserted in column less than or equal to the half beam width.

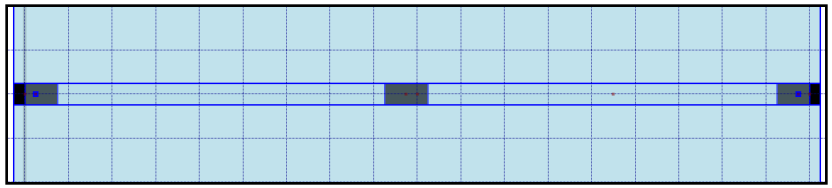


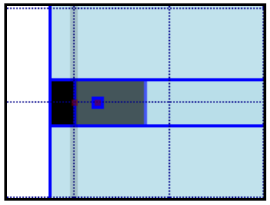


$L_{in} \leq 0.5\ B_{width}$

In case of fixed beams

If the beam drawn to cover the column, it will be act as fixed beam.



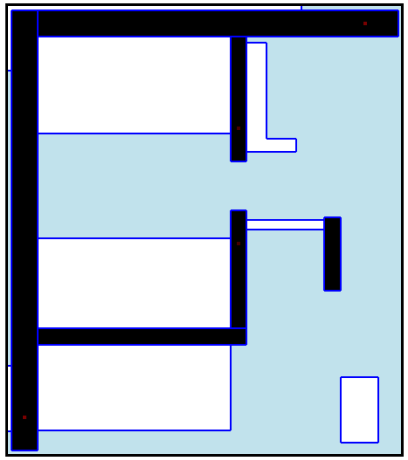


$L_{in} \geq 0.5\ B_{width}$

Assemblies

Assemblies is used if the user want to combine either support or loads together like cores or any shape of shear walls …etc.

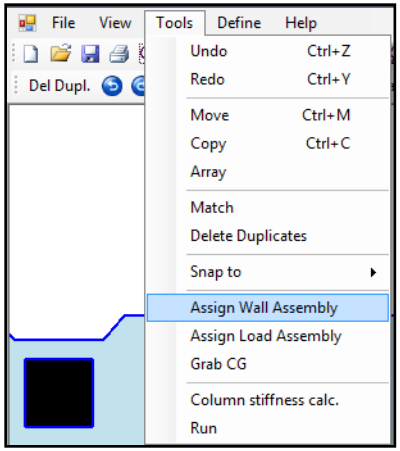
This example for five shear walls and we need to make them deal as one unit.



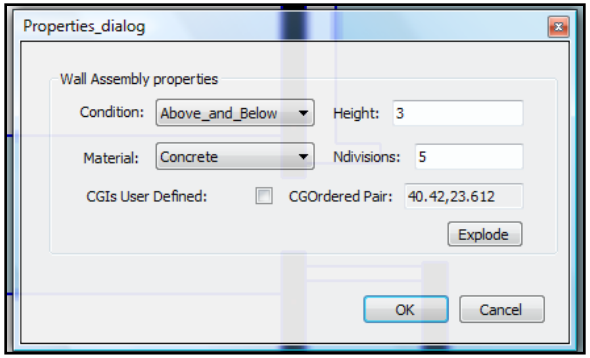
1- Assign them as wall support.

2- Select all walls that need to be combined then click on wall assembly.

OR



If the user press right click on the walls a new dialog similar to wall support opens, but this dialog deals with all shear walls as a one unit.



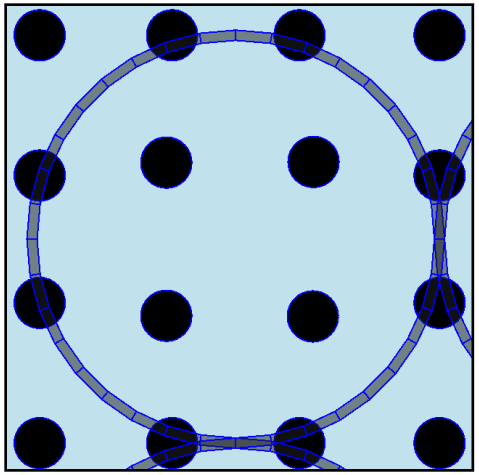
The user can change the condition of the wall, the material type, height of the wall and the number of divisions (Later).

The Explode tab restore the five shear walls again.

Assemblies

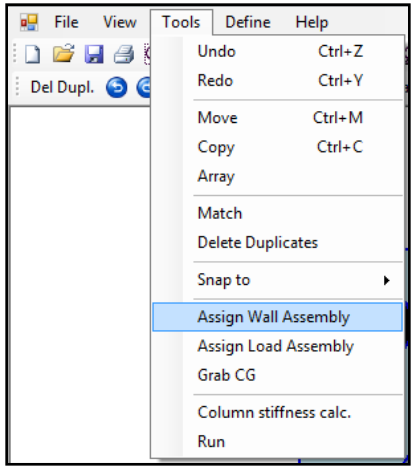
Another examples for Assemblies but in this time we need to combine a load together .

This example silo rested on rafted piles, the silo consists of number of wall loads and we want to combine them together.

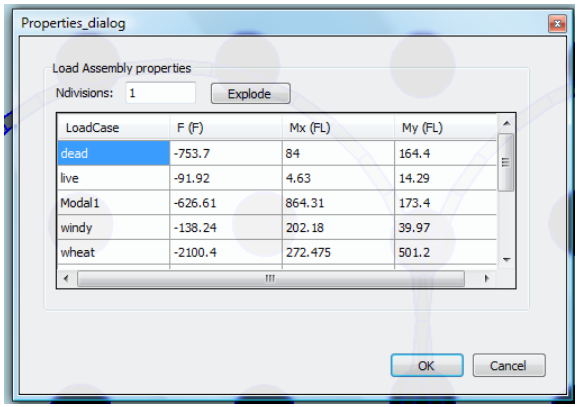


1- Assign them as wall load.

2- Select all walls that need to be combined then click on load assembly.



OR



LoadCase	F (F)	Mx (FL)	My (FL)
dead	-753.7	84	164.4
live	-91.92	4.63	14.29
Modal1	-626.61	864.31	173.4
windy	-138.24	202.18	39.97
wheat	-2100.4	272.475	501.2

If the user press right click on the walls a new dialog similar to wall loads opens, but this dialog deals with all wall loads as a one unit.

the user can insert the load in each load case and the number of division (Later).

The Explode tab restore the wall divisions.

Drop Panel and Multi thickness slab

The user faces slab with multi-thickness or a drop panel in case of modeling flat slab.

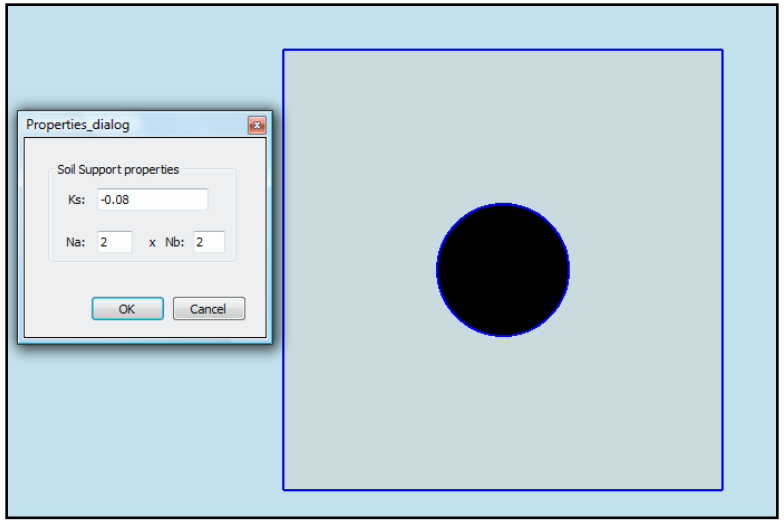
1- Insert the smallest thickness in the slab.

2- Insert the other slabs (drop) as soil support.

3- In the stiffness we insert the thickness but in –ve value.

4- Choose the number of divisions in both directions (Later).

5- Modify the PLCoreMan (Later).



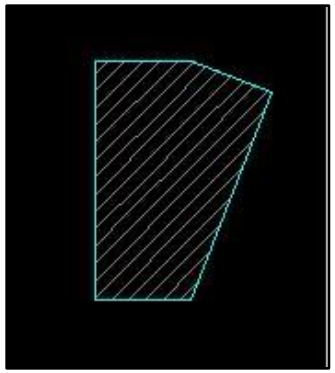
The Drop thickness from zero to 5 times slab thickness according to ECP.

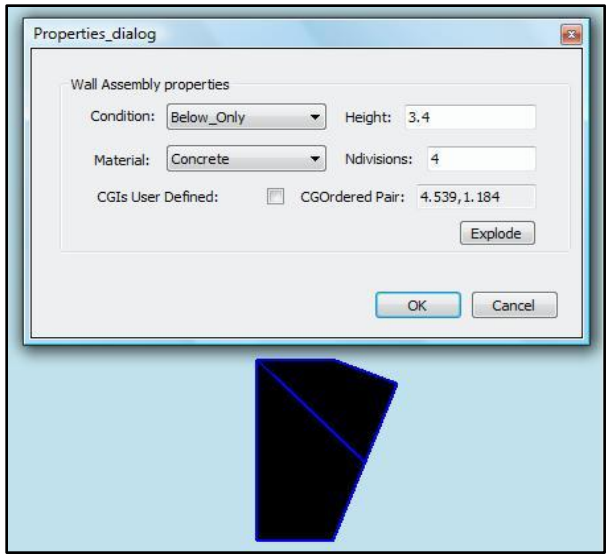
Circular and Non Quadratic Columns

How can we deal with circular column?

Select the center of column then press <u>shift</u> after that press in any place of the outer perimeter of the circle.

What about non quadratic columns?

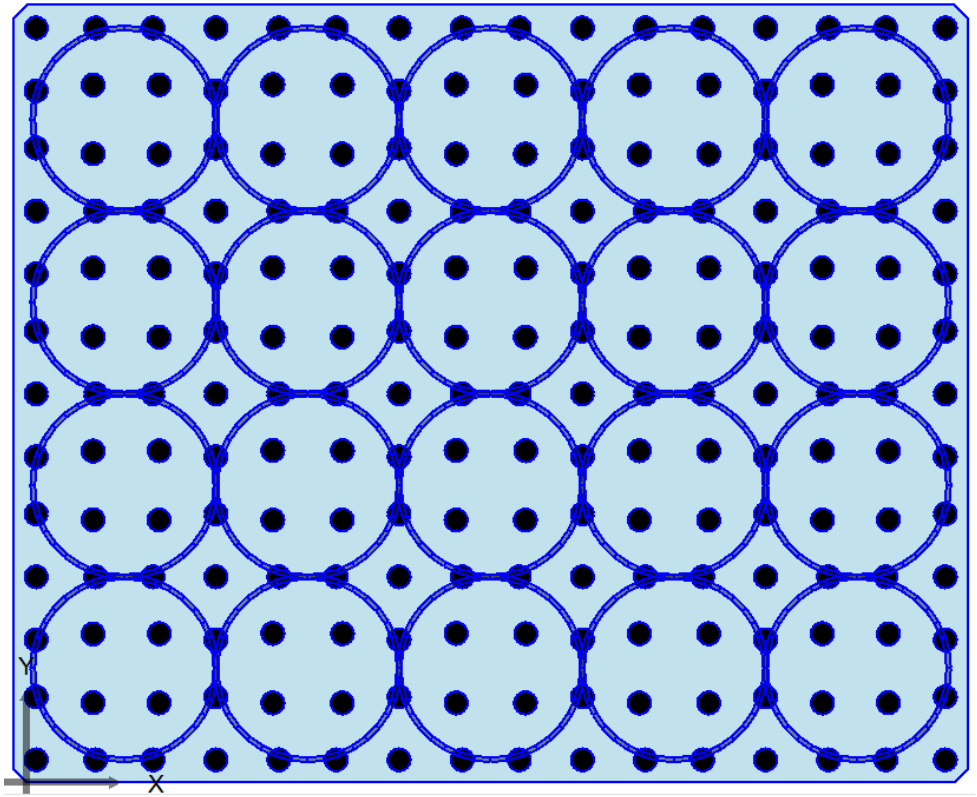




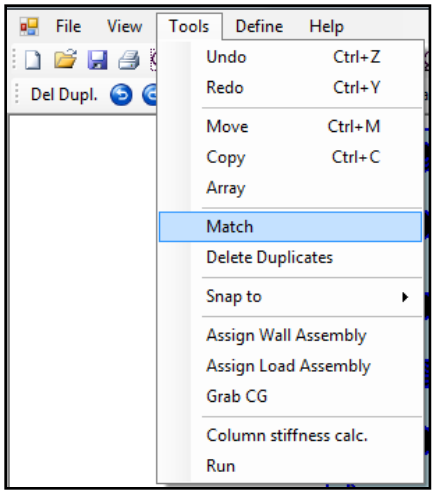
The user must draw them in the AutoCAD as two column each column is a quadratic column, after importing them in the generator file (.Gen) assign them as wall support/loads, then use wall/load assembly.

Many structural elements

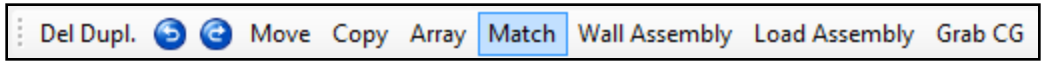
Are the user going to select pile one after another?



Just adjust one pile then select the other piles then match from the adjusted one.

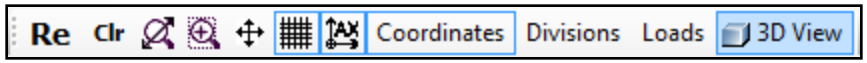


OR

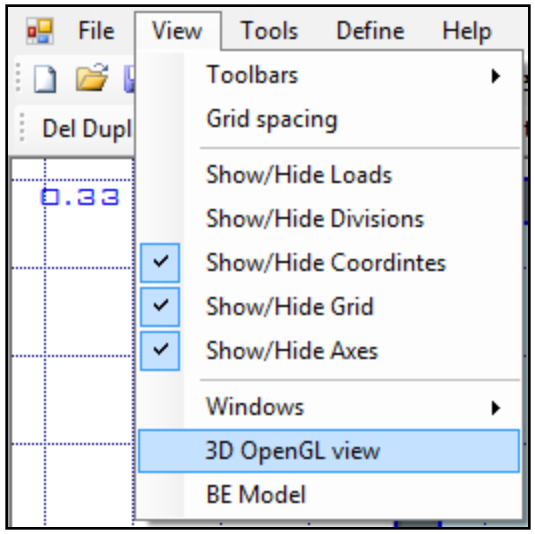


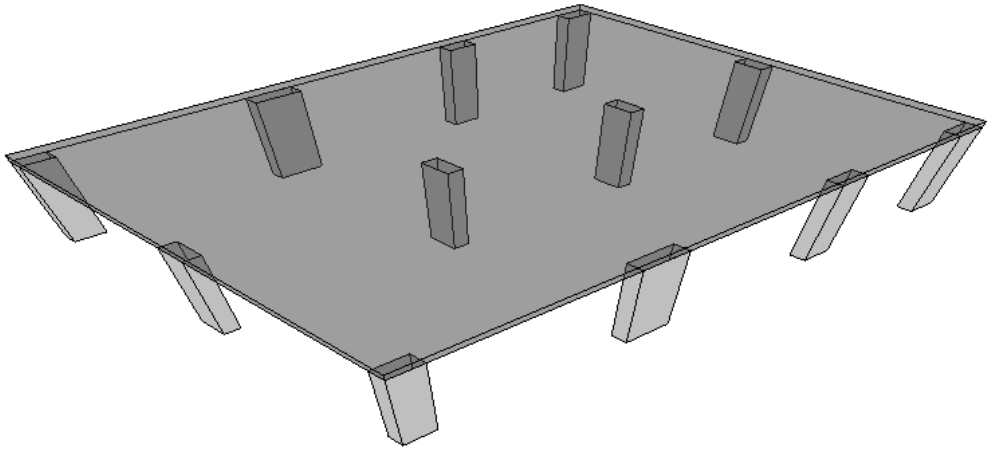
3D Modeling view

Enjoy 3D view of your model after finishing the generator file.

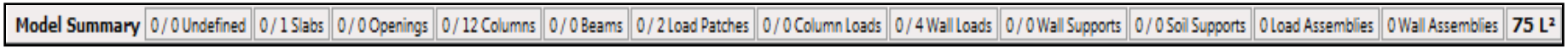


OR





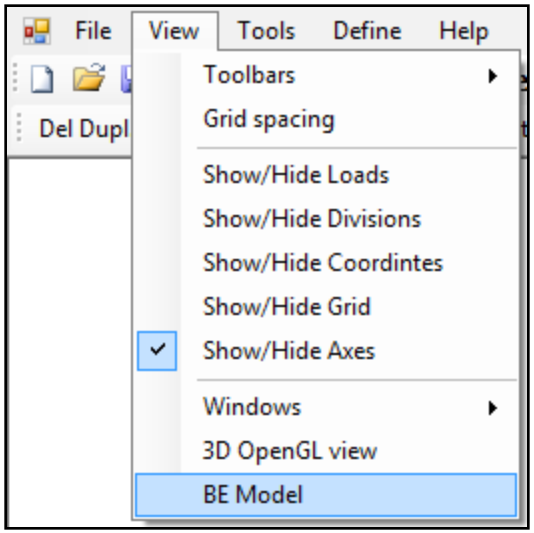
The user can also check that there is not any undefined shape from the lower bar.



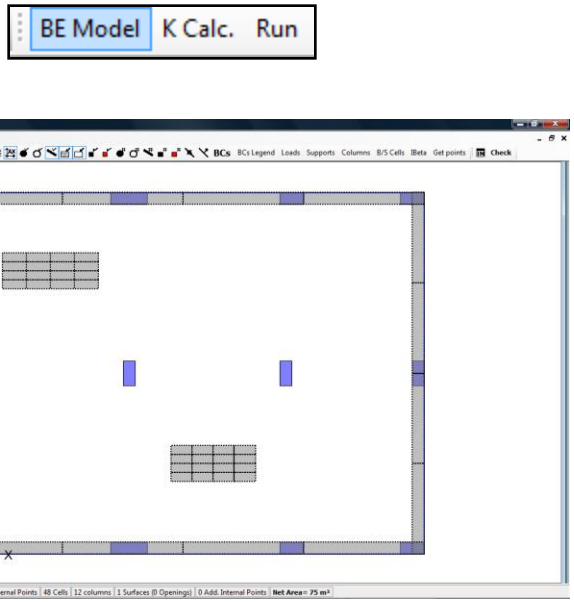
Check model PLView introduction

After finishing all model information the forth step is check modeling before run the analysis.

The purpose of check modeling is to check the boundary nodes and the boundary elements and also we can check about all information that we enter.



OR

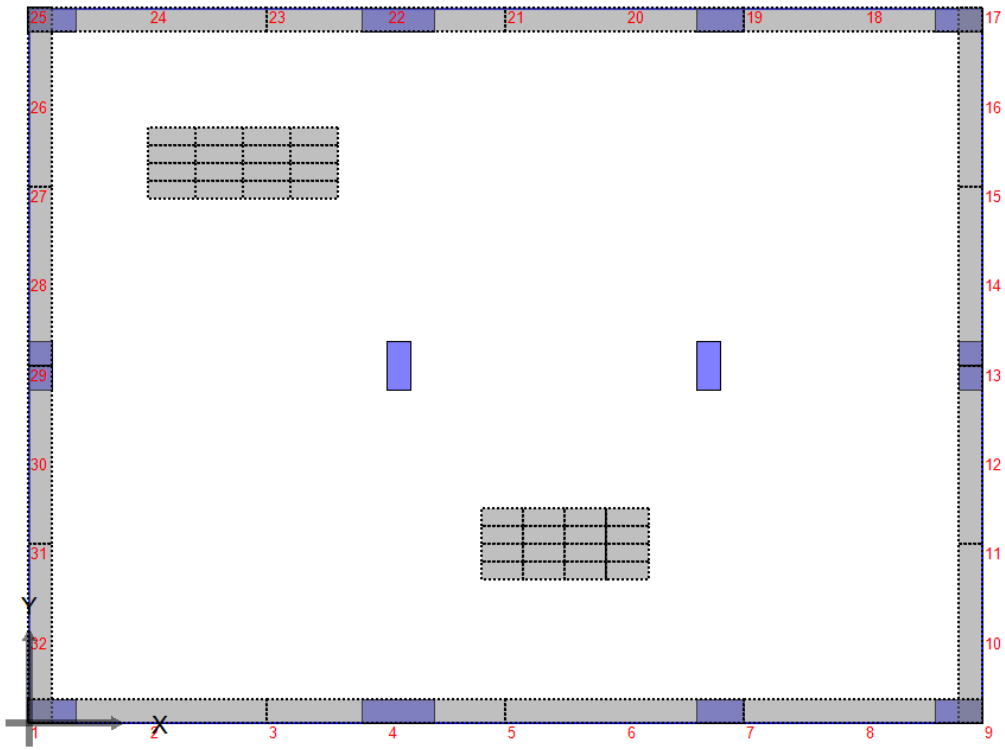


From the PLView we can see the number of division of the wall loads and the load patch.

The model in PLView

At the beginning we should check the node number and element number.

Node number



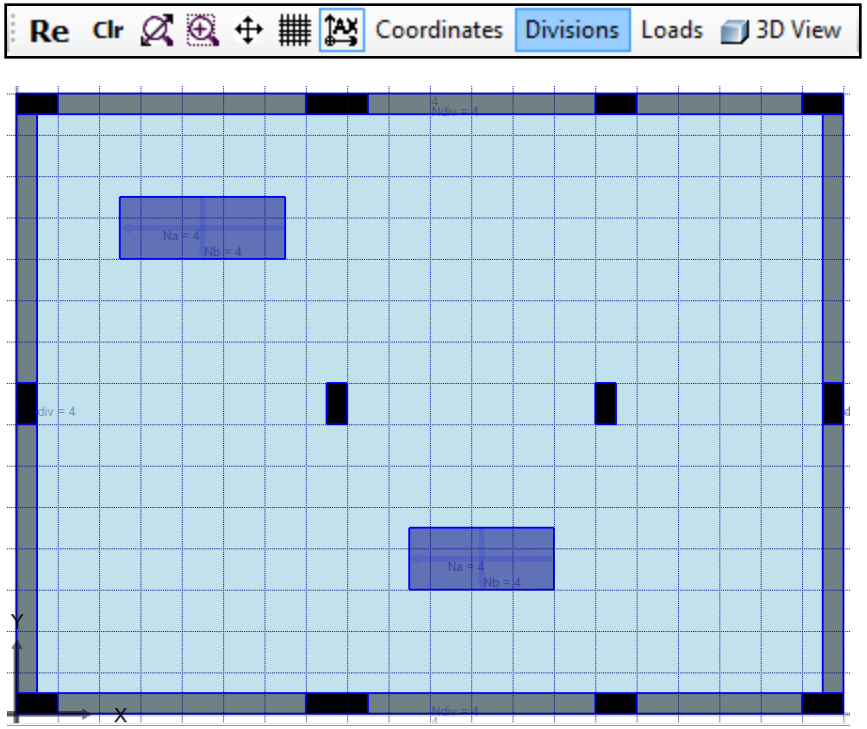
The default number of nodes is 4, we can determine the suitable number of nodes which is the number of inflection of BMD curve.

If two node numbers above each other the model will be unable to run.

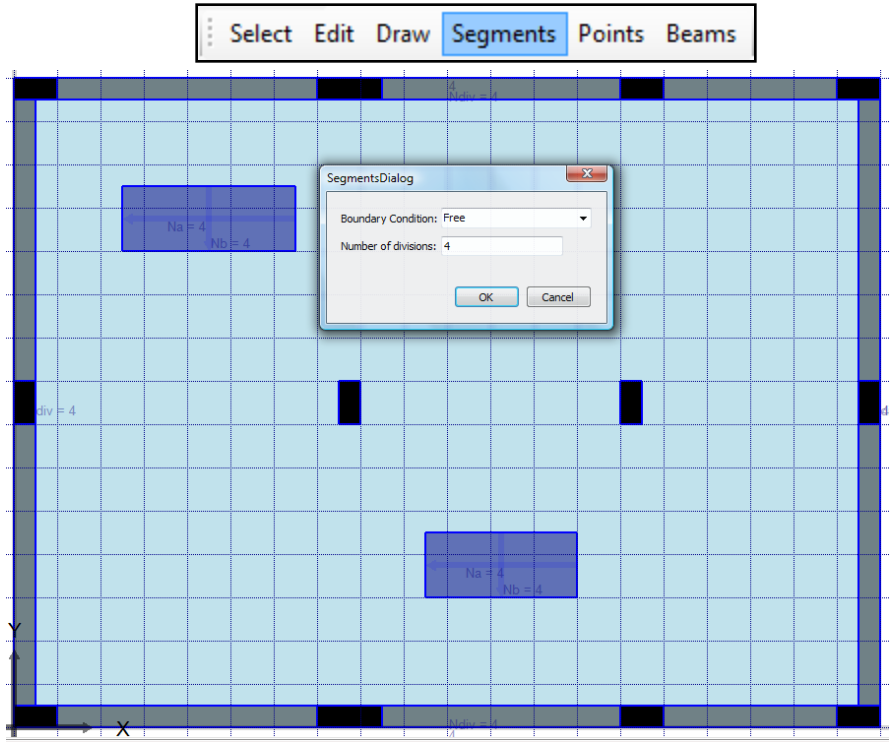
How can we change the number of nodes in the boundary element model?

Changing boundary element nodes

From the Gen file we can show and change the number of nodes.



We can see the number of division for each side, and the number of division for the wall load and the load patch.

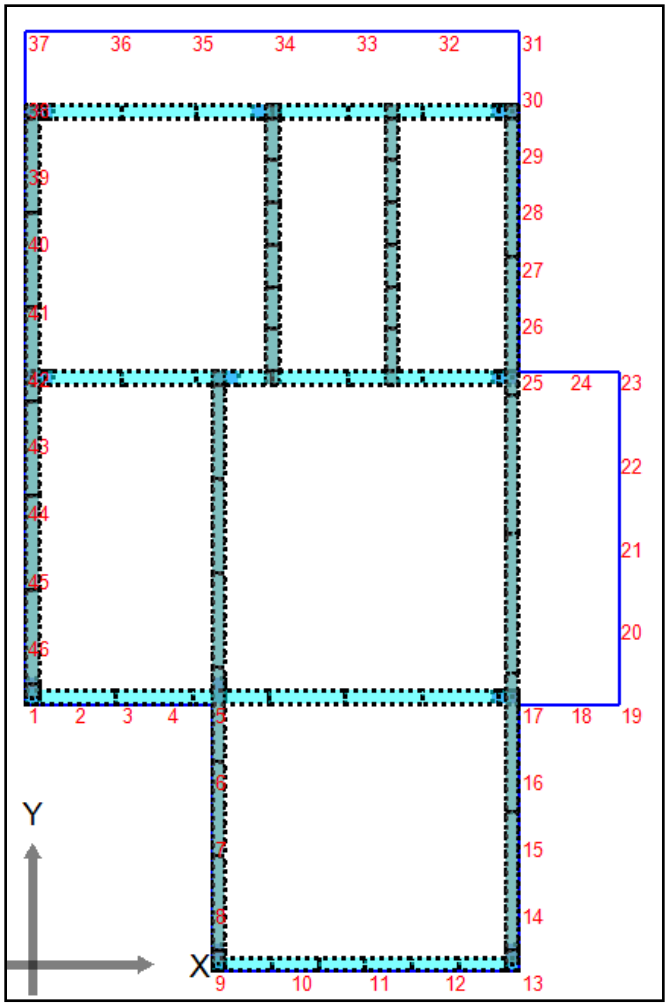


Select the side which need to be changed by left click then right click to open the dialog, which contains the boundary condition (free, hinged, fixed, symmetry about x/y –axes) and the number of divisions.

Beams in PLView

What is the view of beams in the PLView?

In the PLView the user can see the number of division for each beam.



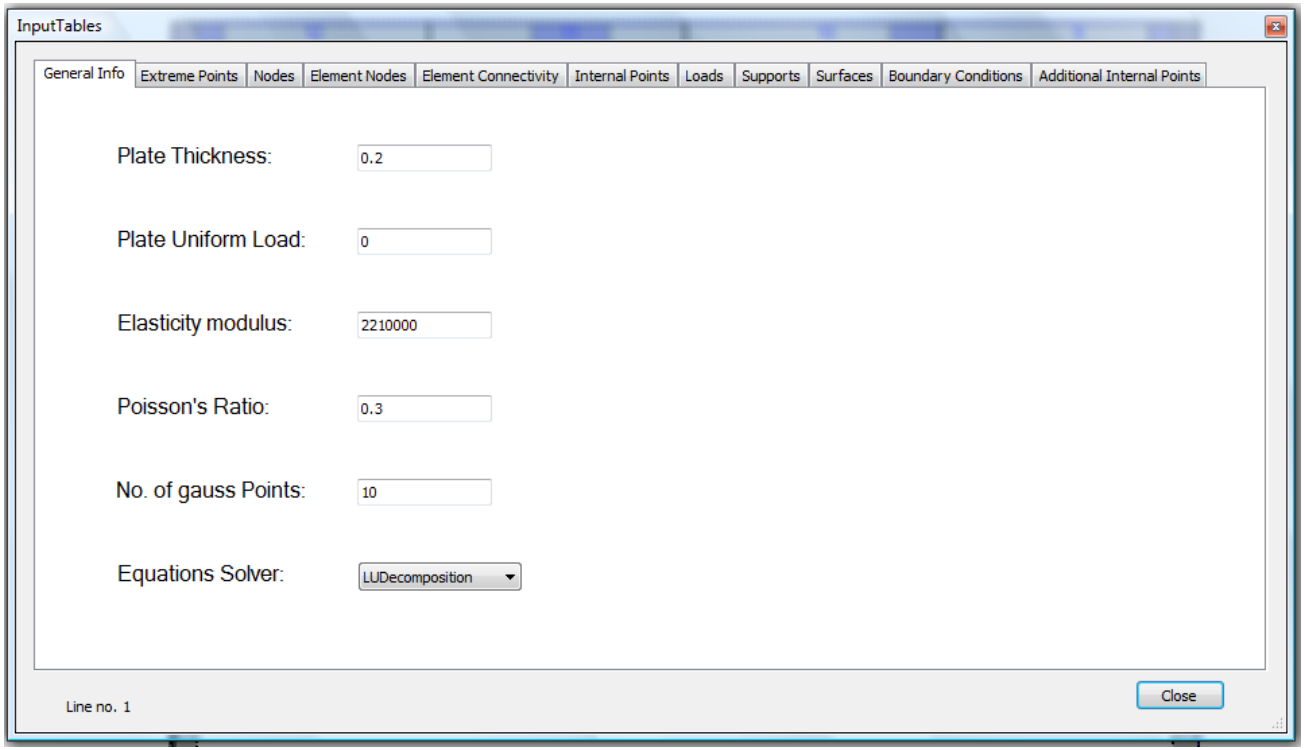
In case the beam color is Dark green, it means that it contains beam own weight.

In case the beam color is light green, it means that it doesn't contain beam own weight.

The model in PLView

In case of there is a problem in running or illogic results due to user mistakes, We can check the input tables.

Input table

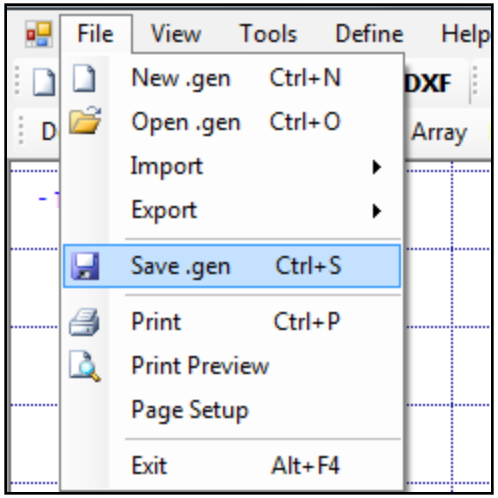


This table contains all the information for the file like General information that user inserts, number of nodes and it's coordinates, element connectivity, loads values, support elements and it's stiffness.

Run Model (PLCoreMan Introduction)

Finally the generator file is now ready for running.

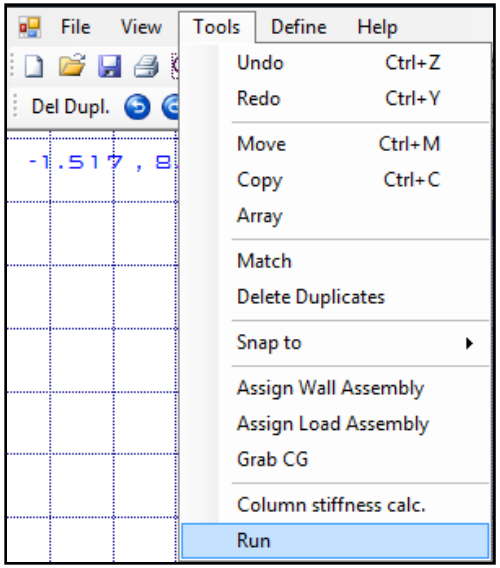
Save the generator file (Gen).



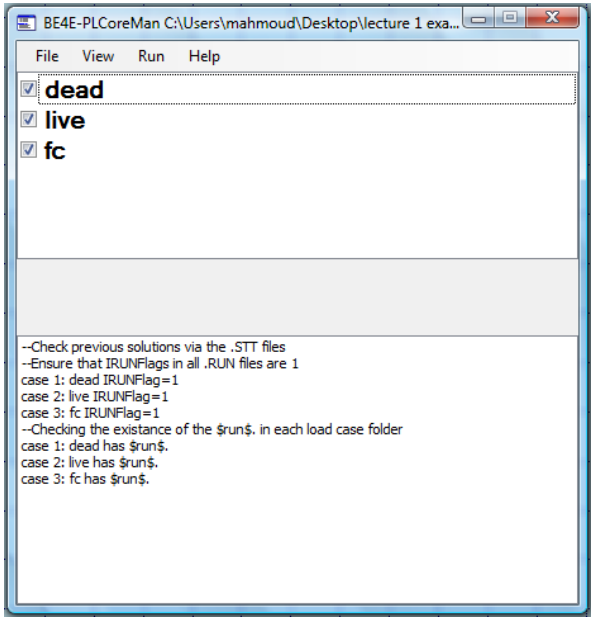
OR

Run the generator file (Gen).

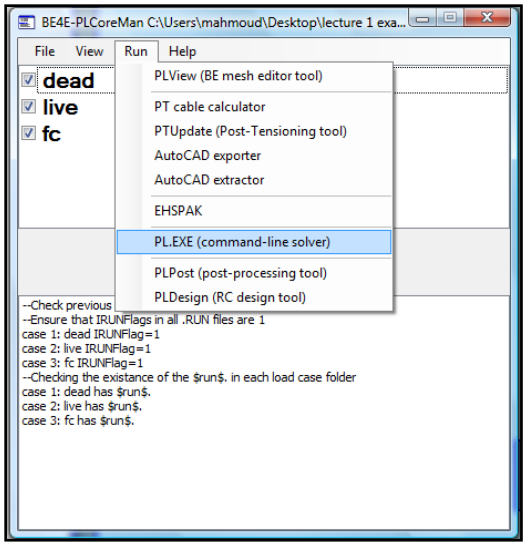
OR

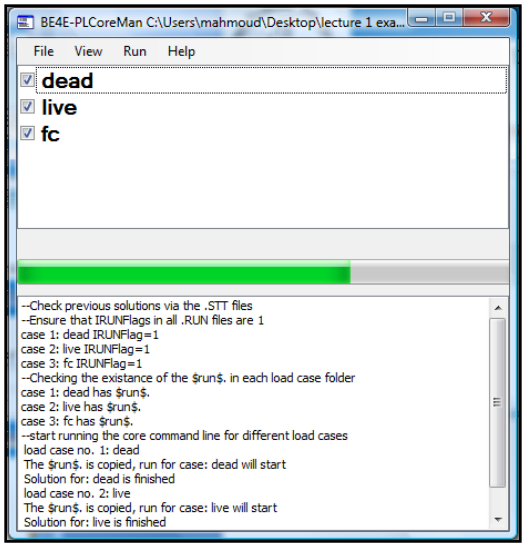


Save load case file (.LC) to open PLCoreMan .

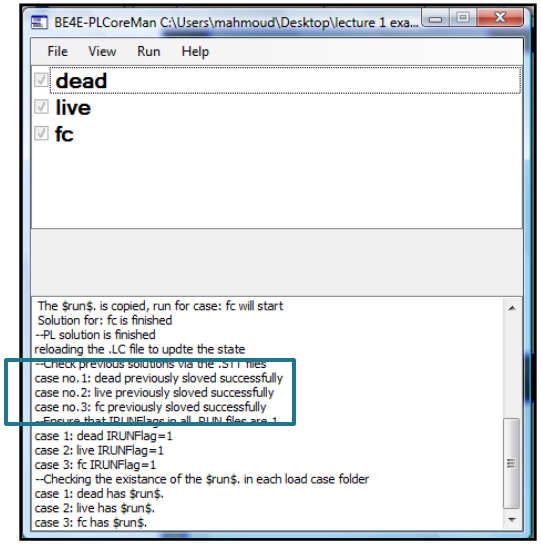


To run the model go to run menu then PL.EXE.





The user should see in the text window that the load cases run successfully.



If the load cases didn't run successfully we have to check our generator model.

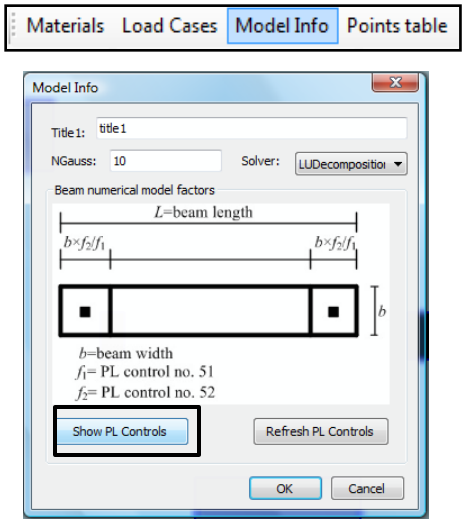
PLControls

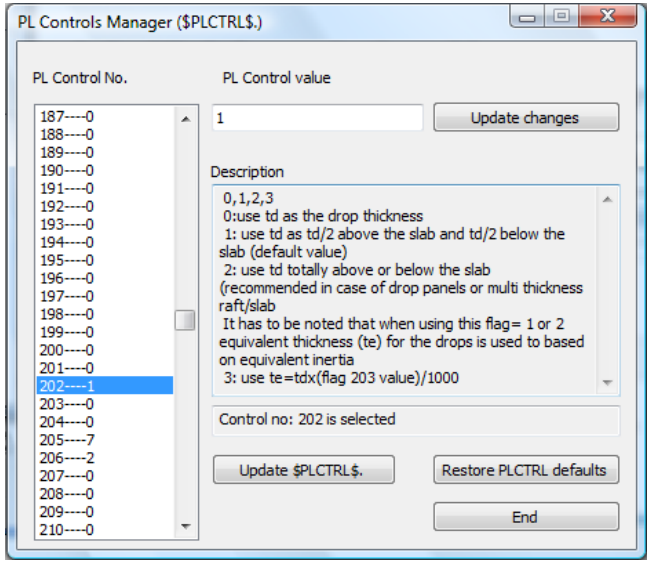
PLControls is very important window in case of Drop Panel or Multi thickness slab.

- Modification in PLControls should be before running the model.

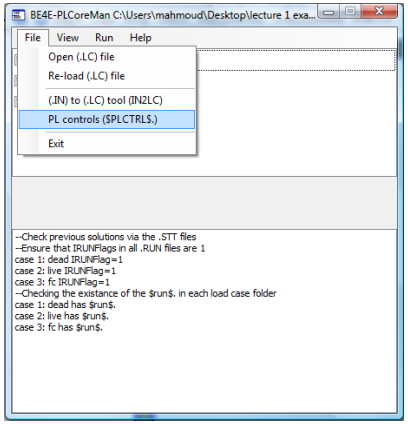
- The user can change the PLControls by two ways either from generator file or PLCoreMan.

From Gen file





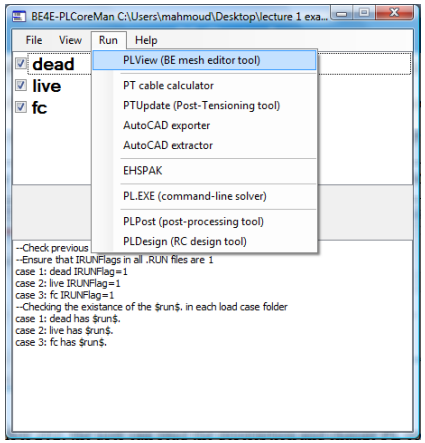
From PLCoreMan



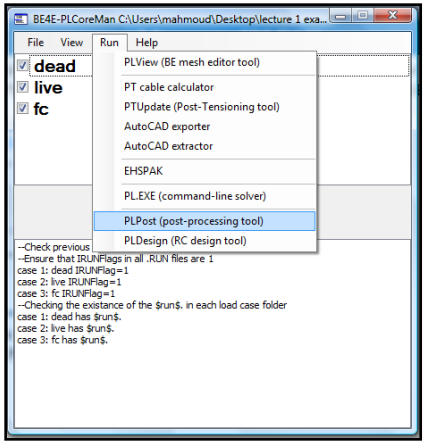
From PL Control number 202, the user can read the description and change PL control value, then click on update $PLCTRL$ to save the update, after that the user can use the model .

Manager of PLPAK

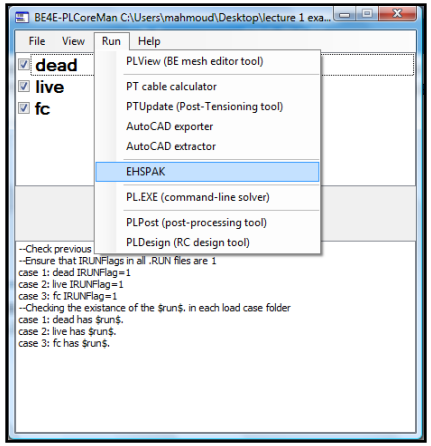
PLCoreMan is the Manger of PLPAK we can transfer between file of PLPAK (PLView & PLPost) or between Packages (PLDesign & EHSPAK).



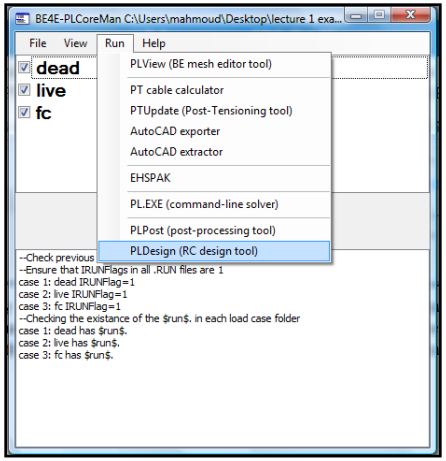
If there is any problem we can check the model from PLView.



After running the user can move to PLPost to view results.



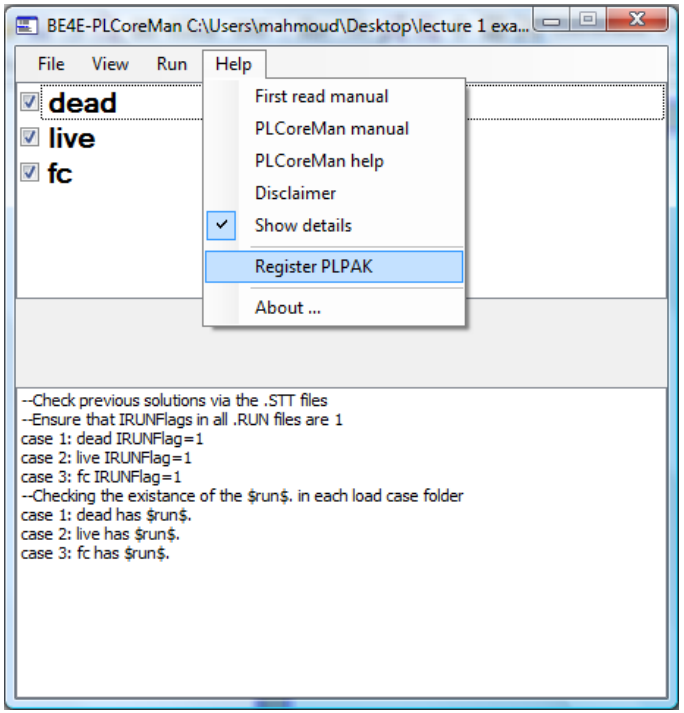
The user can use EHSPAK before running the model.

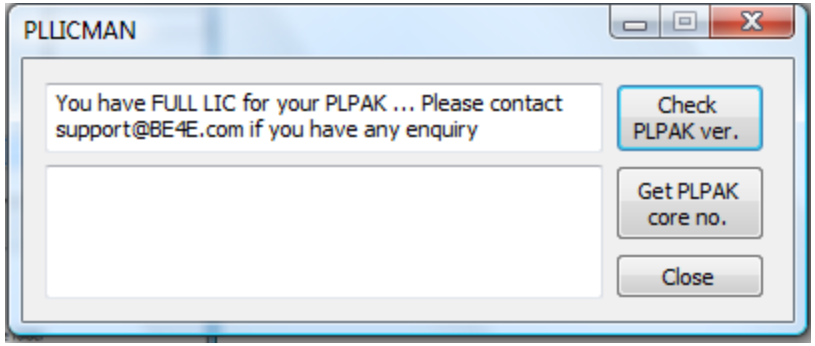


The user can use PLDesign for design the model after running.

Check of License

PLCoreMan can check the user license is valid or not.





Check PLPAK version from help menu then Register PLPAK and from check PLPAK version.

Another function for the PLCoreMan is after installing package is to get PLPAK core number.