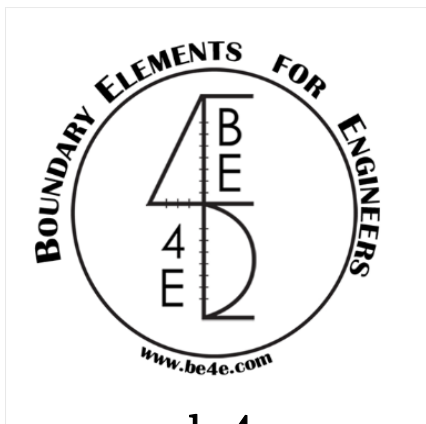




Cairo University

PLPAK: BEM based software

Presented by
Prof. Youssef F. Rashed
Dr. Ahmed Fady Farid



www.be4e.com



هيئة تمويل العلوم والتكنولوجيا والابتكار
Science, Technology & Innovation Funding Authority



CUFE-BE

Cairo University
Faculty of Engineering
Boundary Elements Group

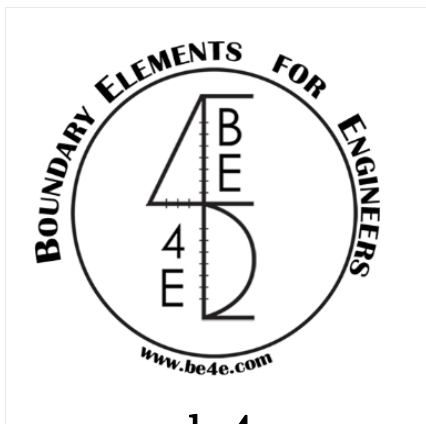
2021



Cairo University

PLPAK: BEM based software

Presented by
Prof. Youssef F. Rashed
Dr. Ahmed Fady Farid



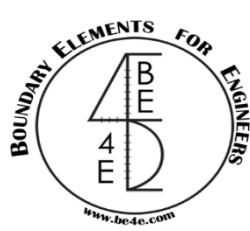
www.be4e.com



Cairo University
Faculty of Engineering
Boundary Elements Group

2021

CUFE-BE



www.be4e.com



Cairo University

Table of contents

- 1. Introduction**
- 2. Basic package (PLPAK Basic)**
- 3. One floor package (BIM-PLPAK)**
- 4. Foundation Package (FoundPAK)**
- 5. Fixed base Package (FBPAK)**
- 6. Post-tension Package (PTPAK)**
- 7. Dynamic package (DynPAK)**
- 8. Overall building package (OBPAK)**
- 9. 4D and 5D analysis**
- 10. Conclusions**

CUFE-BE



www.be4e.com



Cairo University

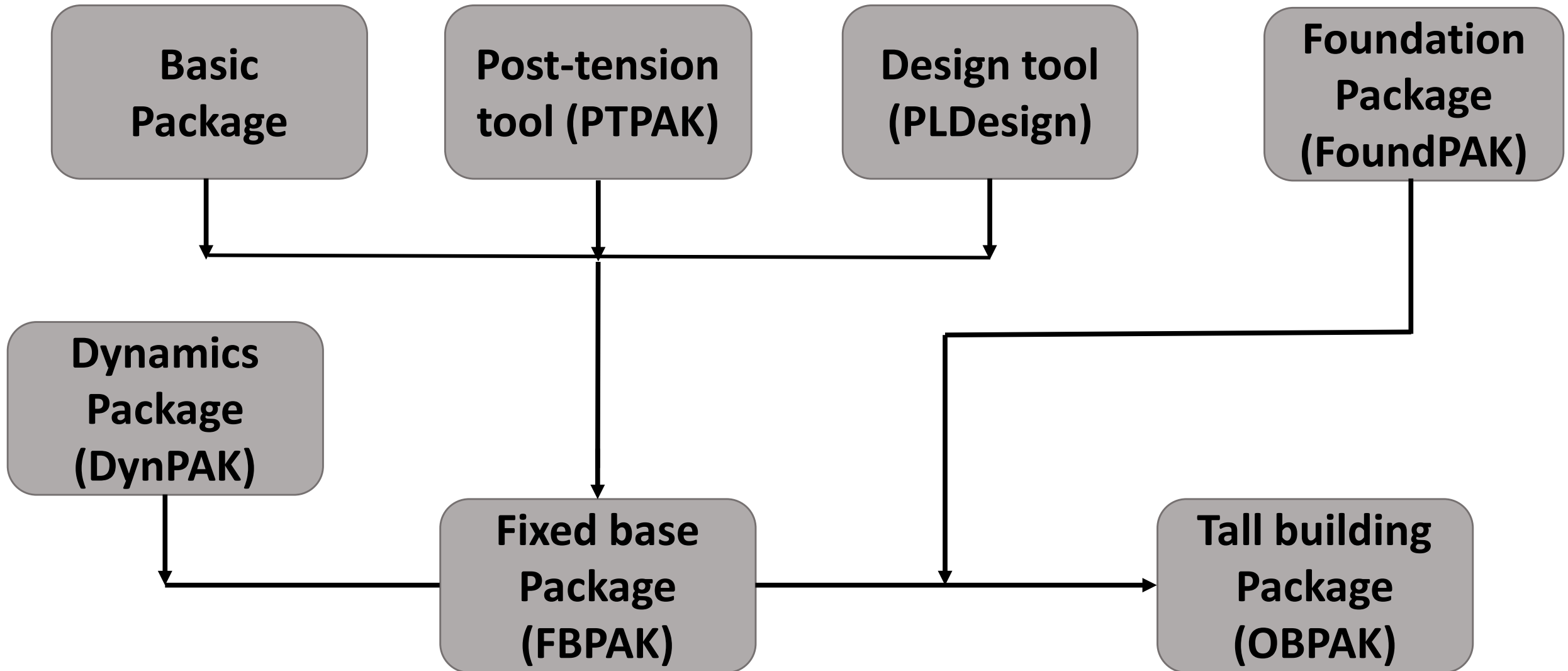
Table of contents

- 1. Introduction**
2. Basic package (PLPAK Basic)
3. One floor package (BIM-PLPAK)
4. Foundation Package (FoundPAK)
5. Fixed base Package (FBPAK)
6. Post-tension Package (PTPAK)
7. Dynamic package (DynPAK)
8. Overall building package (OBPAK)
9. 4D and 5D analysis
- 10. Conclusions**

What is the PLPAK?

- PLPAK (plate analysis package) is a structural analysis software package for plate bending structures based on the boundary element method for shear deformable plate bending theory.
- The PLPAK-Basic is a software package for structural analysis of building slabs and foundations.
- 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.
- The PLPAK has also been verified by papers published in highly ranked journals. And its results are verified by several other analytical and reliable numerical methods, as well as proving to be much quicker at solving irregular large practical models.

1. Introduction



CUFE-BE



www.be4e.com

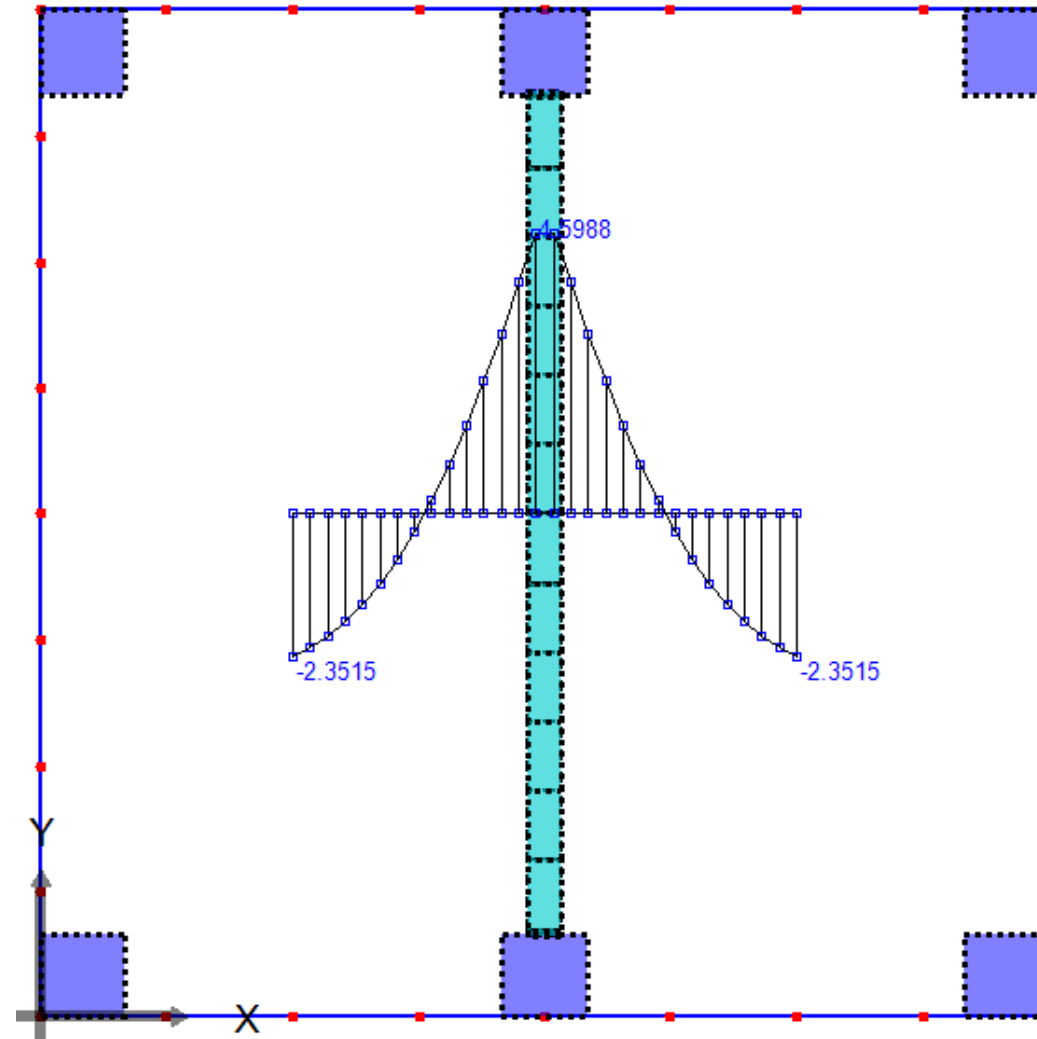
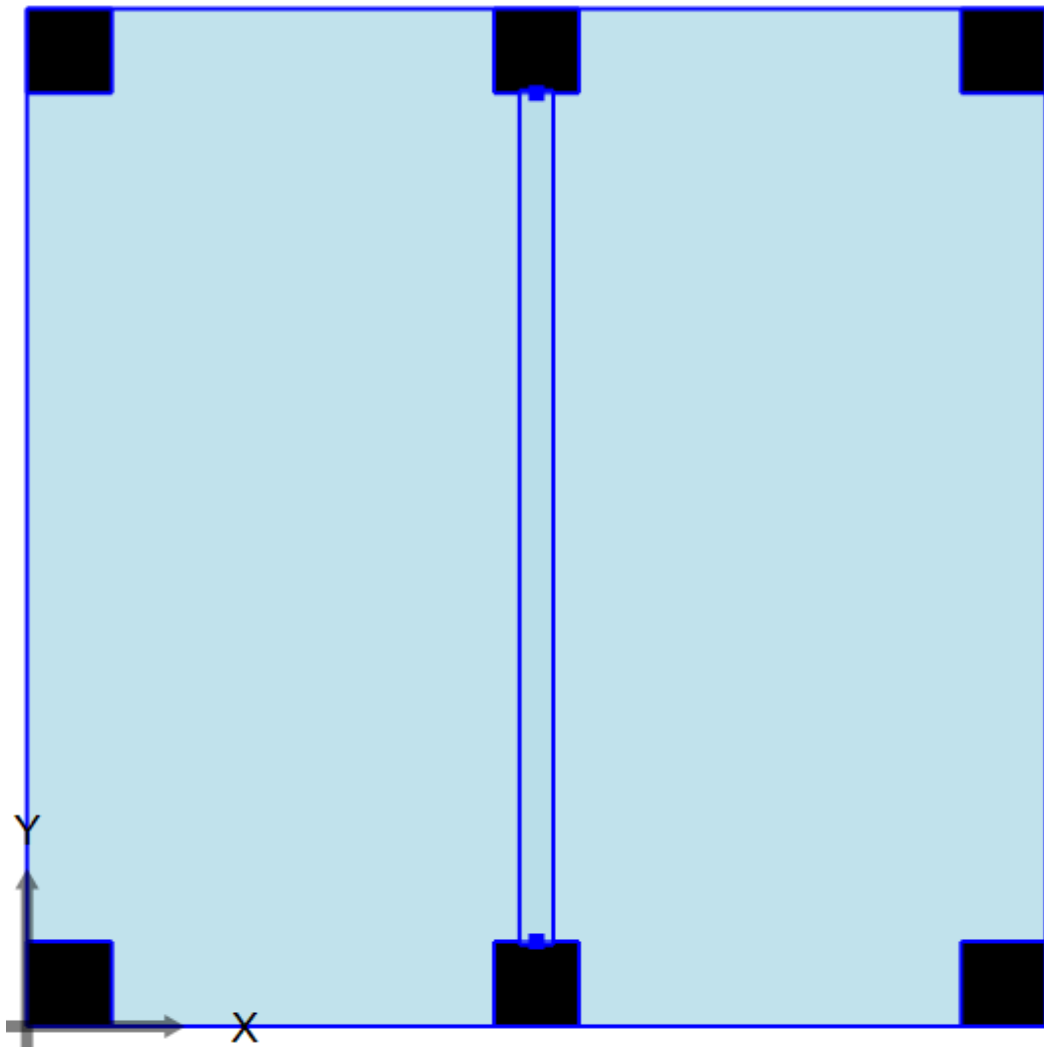


Cairo University

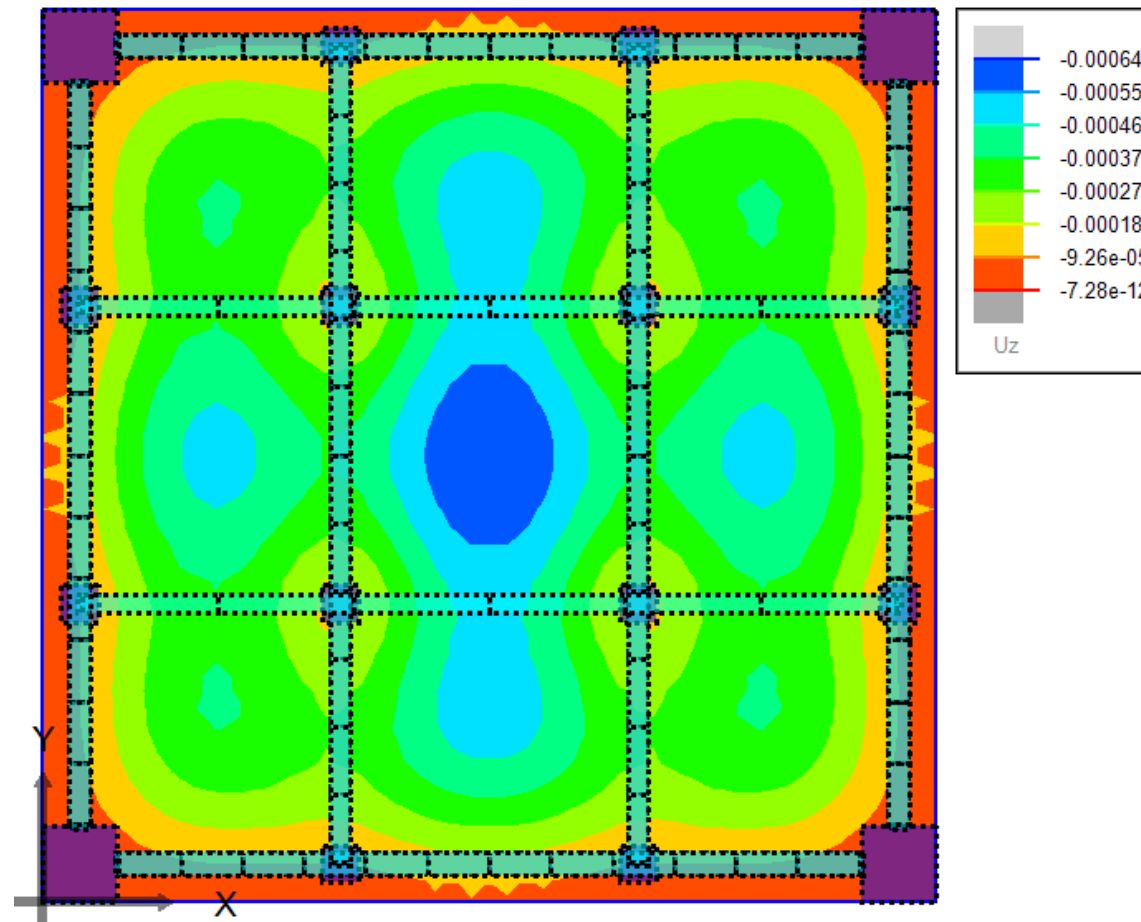
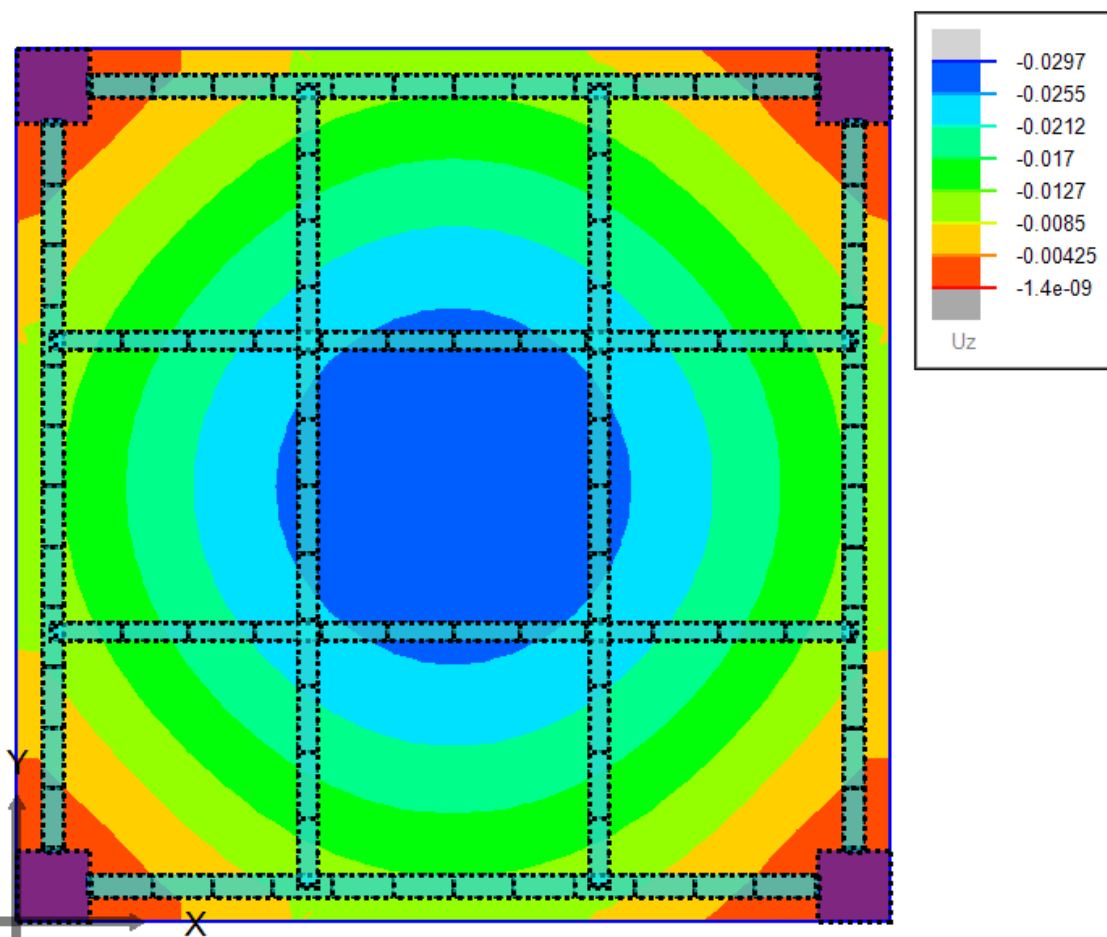
Table of contents

- 1. Introduction**
- 2. Basic package (PLPAK Basic)**
3. One floor package (BIM-PLPAK)
4. Foundation Package (FoundPAK)
5. Fixed base Package (FBPAK)
6. Post-tension Package (PTPAK)
7. Dynamic package (DynPAK)
8. Overall building package (OBPAK)
9. 4D and 5D analysis
- 10. Conclusions**

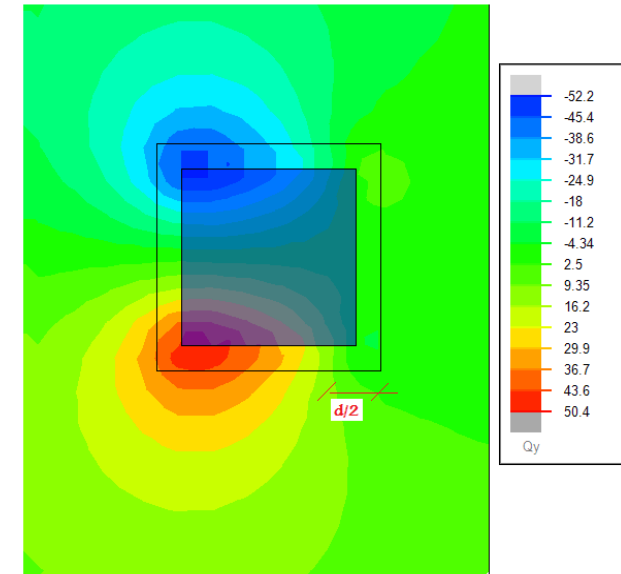
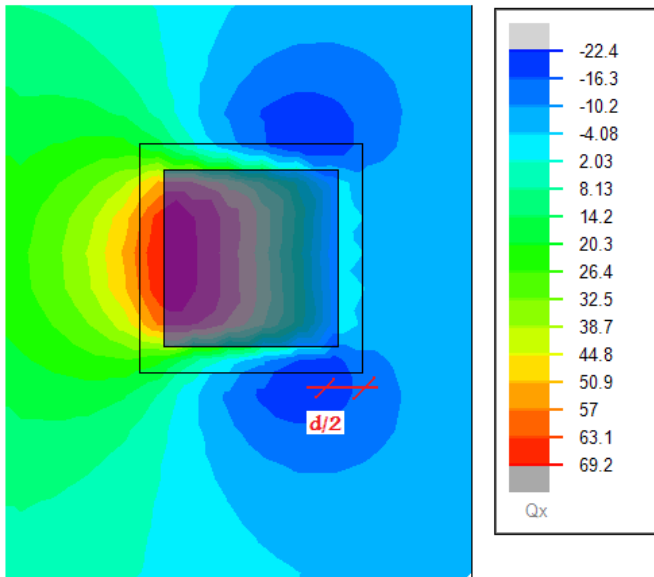
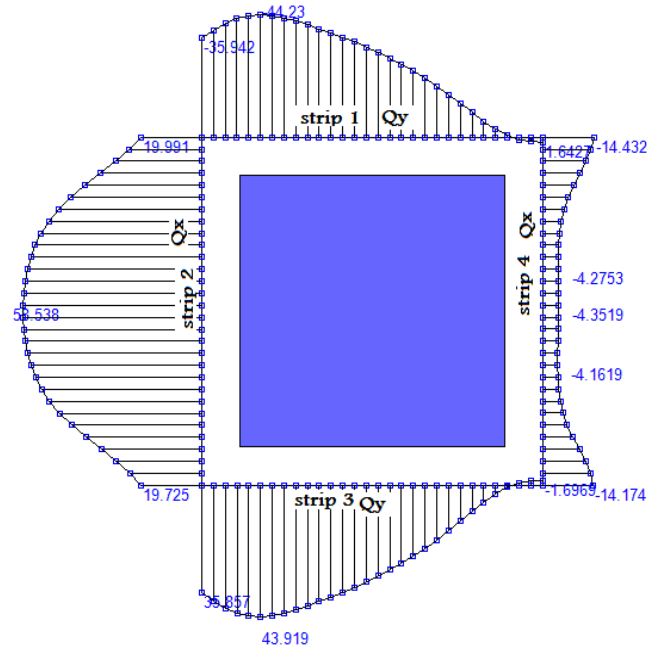
2. Basic package (PLPAK Basic)



2. Basic package (PLPAK Basic) www.be4e.com

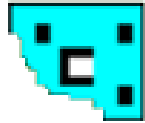


2. Basic package (PLPAK Basic) www.be4e.com

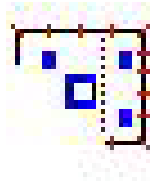


PLPAK Basic – package components:

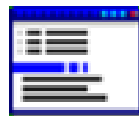
1- Generator (PLGen).



2- Boundary element numerical model viewer (PLView).



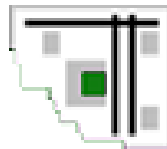
3- Manager (PLCoreMan).



4- Post-processing (PLPost).



5- Design tool (PLDesign)



CUFE-BE



www.be4e.com



Cairo University

A. PLGen – Model generator

B. PLView – Numerical model

C. PLCoreMan – Manager and solver

D. PLPost – Post processing

E. PLDesign – Design tool

F. PLPAK modelling capabilities

CUFE-BE



Cairo University

The PLGen is responsible for generating the model showing every structural element.

Generating the model is in main three steps:

- 1. Edit model information**
- 2. Build model and define its elements**
- 3. Edit the boundary element divisions**

1. Edit model information

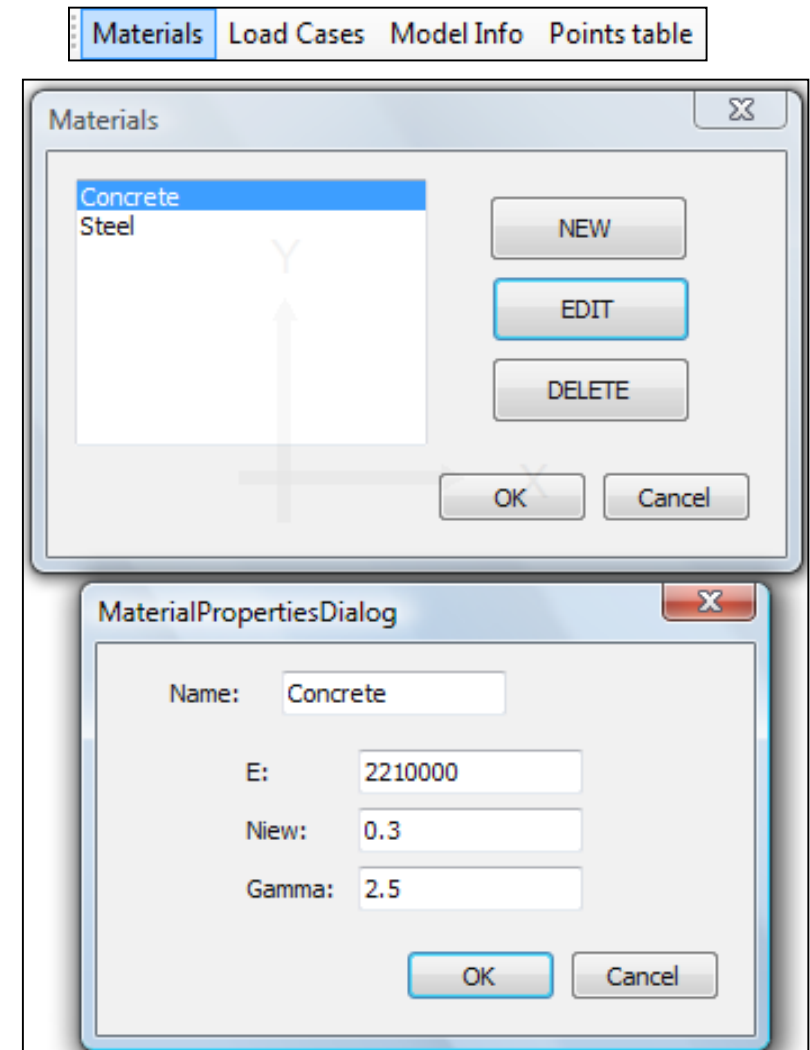
In the model information we will define material properties from material tab



OR

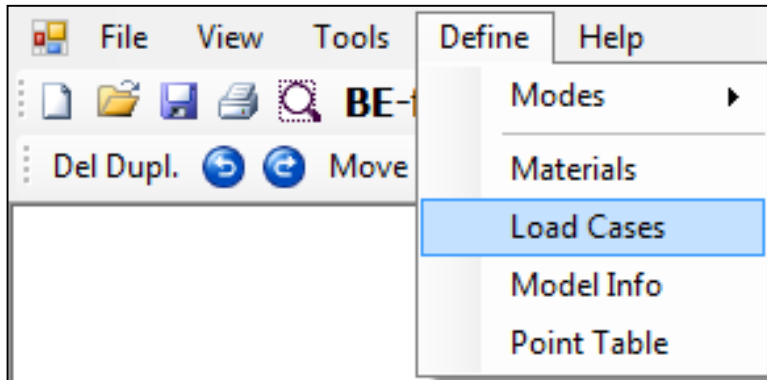
The user can define the properties of the construction material such as Young's modulus, Poisson's ratio and Gamma of the materials.

It has to noted that PLPAK is a dimensionless software, that is mean that the user should input the information in a compatible units.

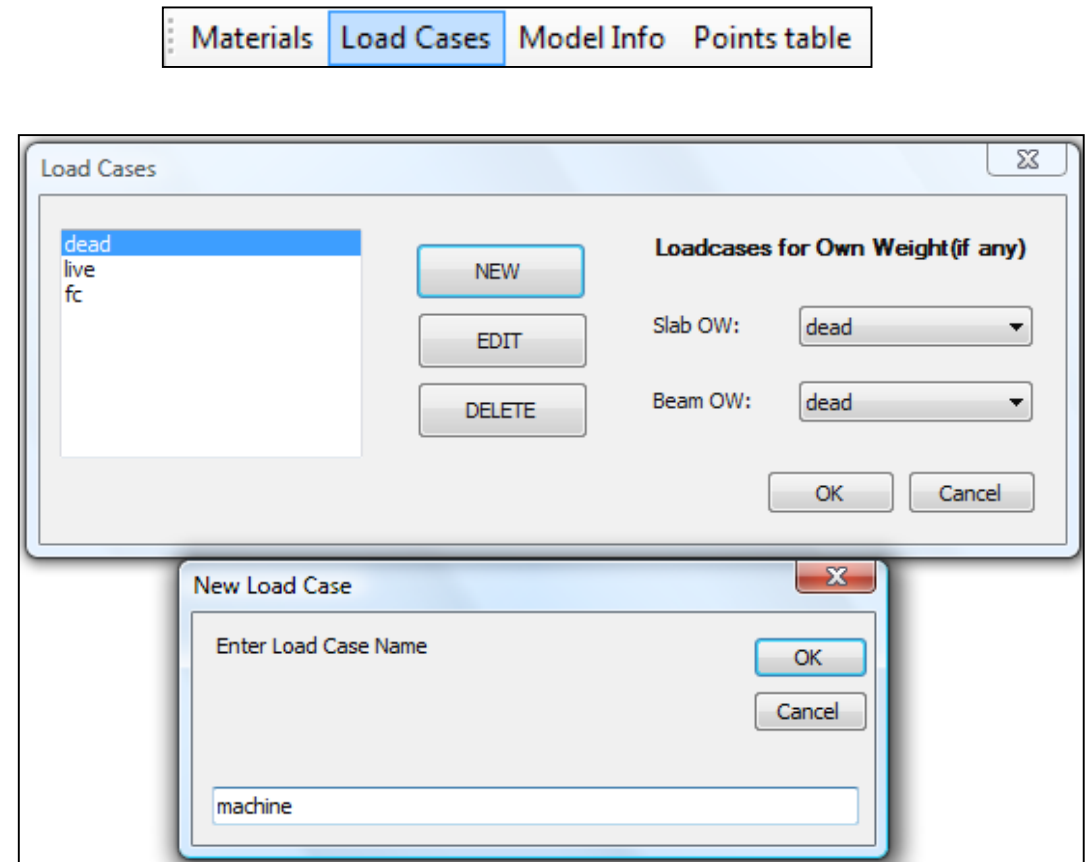


1. Edit model information

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



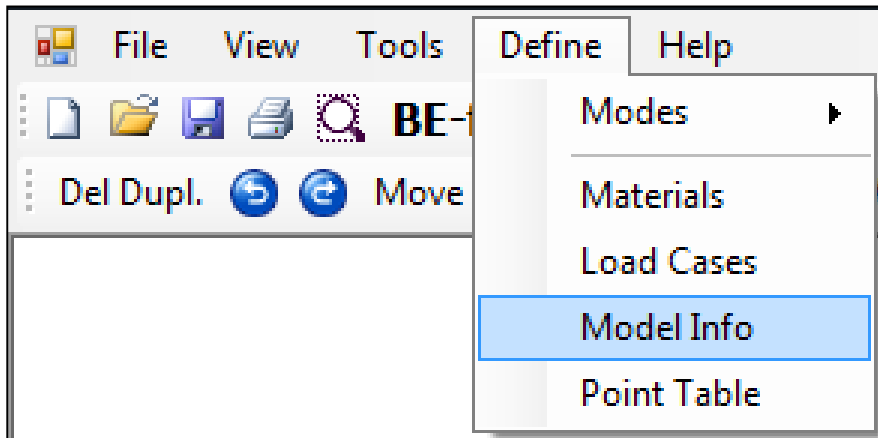
OR



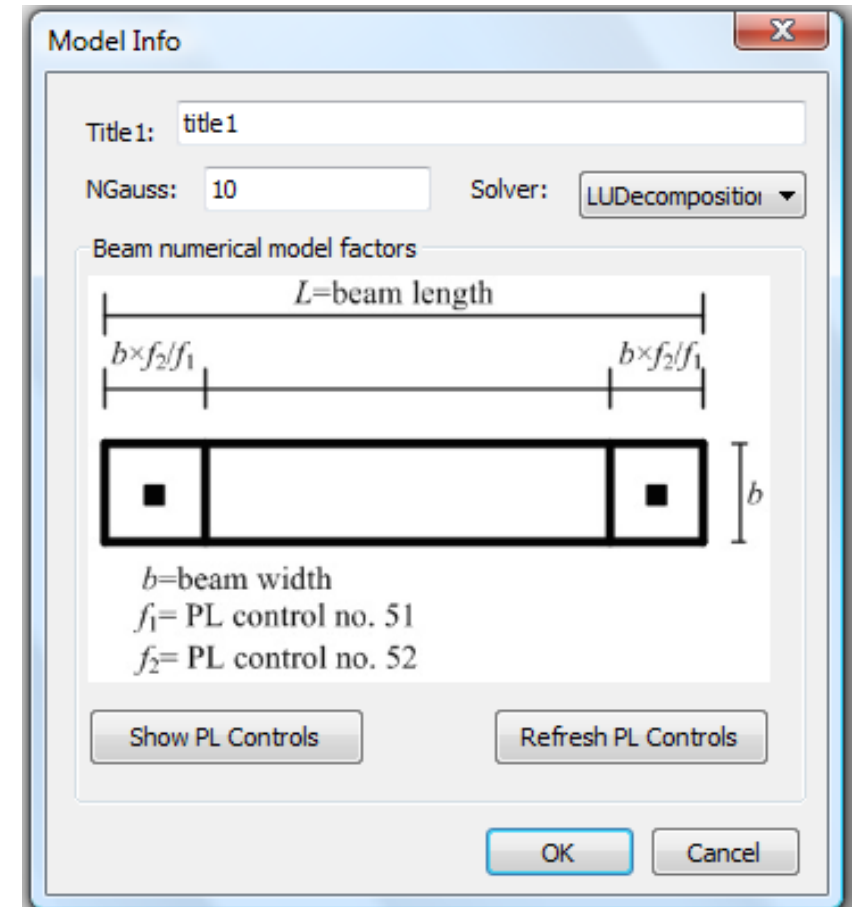
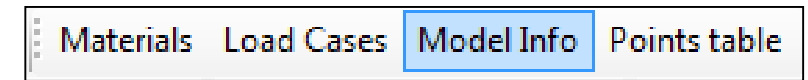
The user can define different load cases and which one can include the slab and beam own weight.

1. Edit model information

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 even number.
- The user can change the type of solver either LU Decomposition, Gauss Elimination, or GPU solver.
- Also the user can change the PL Controls.

1. Edit model information

PL Controls are 400 Ctrl for:

- Modelling aspects.
 - Printing out data.
 - Solver and numerical model.
-
- PLPAK gas its default PL Controls.
 - In case of changing any control you need to update \$PLCTRL\$ file form **Update \$PLCTRL\$** button.
 - **Restore PLCTRL defaults** button restore them to the default values.

www.be4e.com

PL Controls Manager (\$PLCTRL\$.) _Cairo University

PL Control No.	PL Control value
1--1	<input type="text"/>
2--1	
3--1	
4--1	
5--1	
6--1	
7--1	
8--1	
9--1	
10--1	
11--1	
12--1	
13--1	
14--1	
15--1	
16--1	
17--1	
18--1	
19--1	
20--1	
21--3	
22--4	
23--1	
24--4	

Description

Controls are loaded from \$PLCTRL\$.

2. Build model and define its elements www.be4e.com

The user can insert the model by three ways

Drawing Model from PLGen

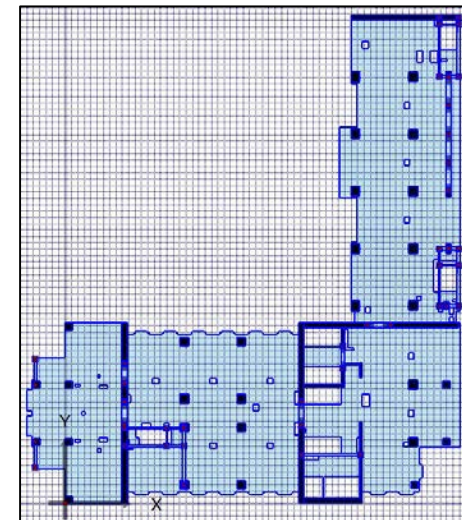
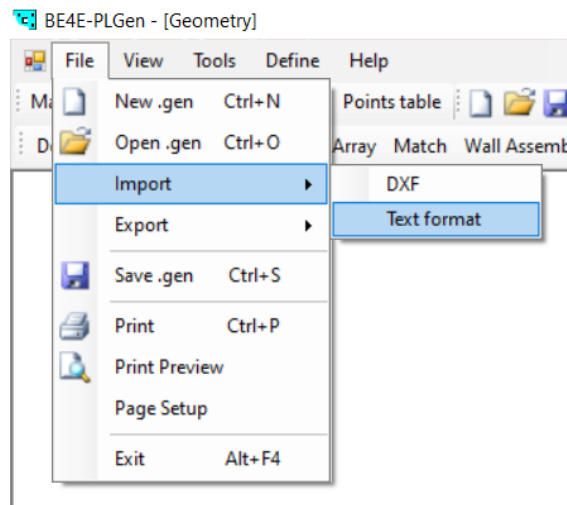
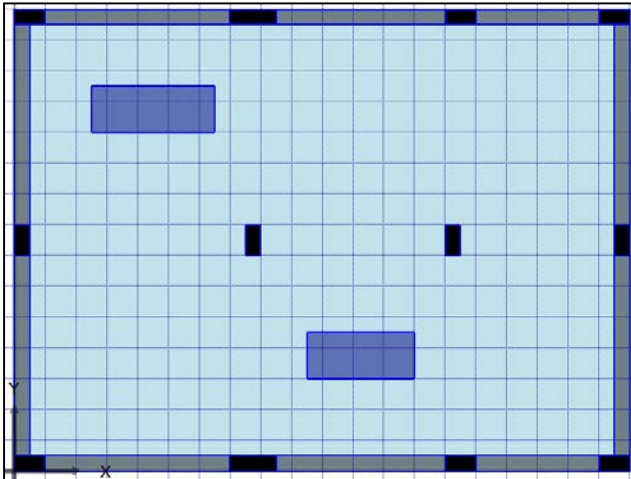
This option is useful for small models.

Importing Model from text files

This option is useful for developers.

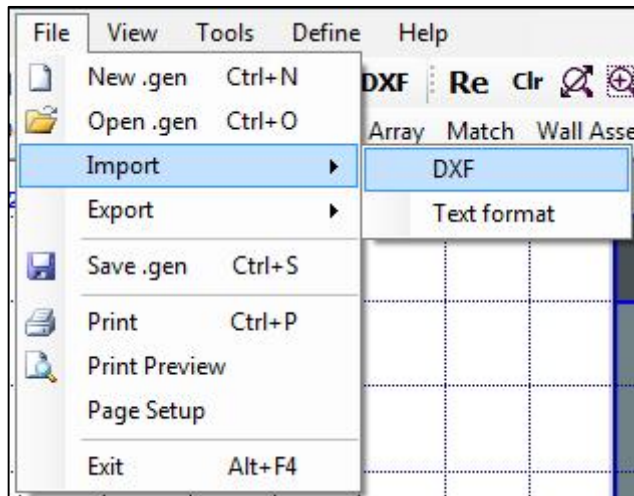
Importing (.Dxf) file (AutoCAD file)

This option is useful for large models or detailed models.

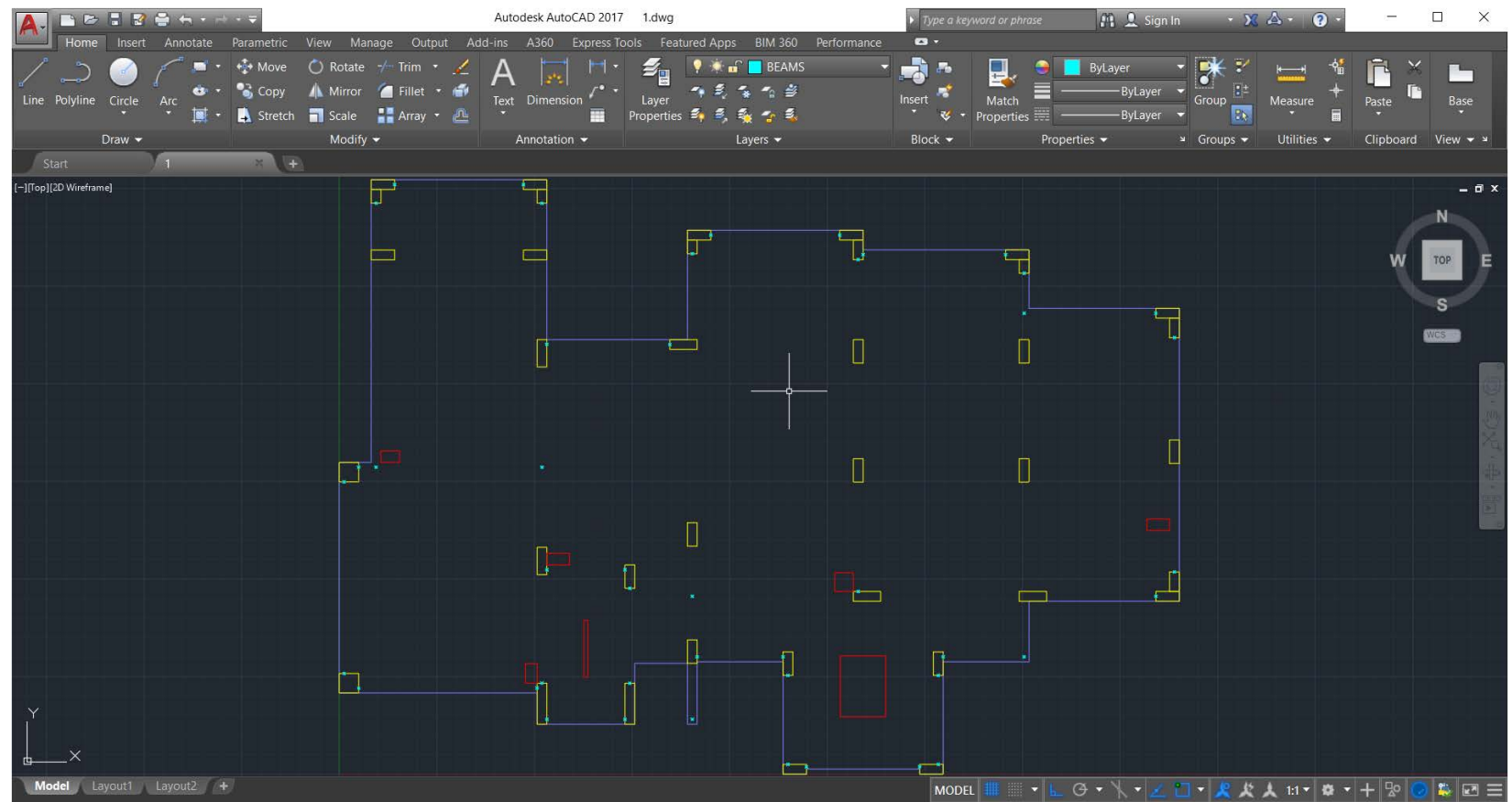


2. Build model and define its elements www.be4e.com

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



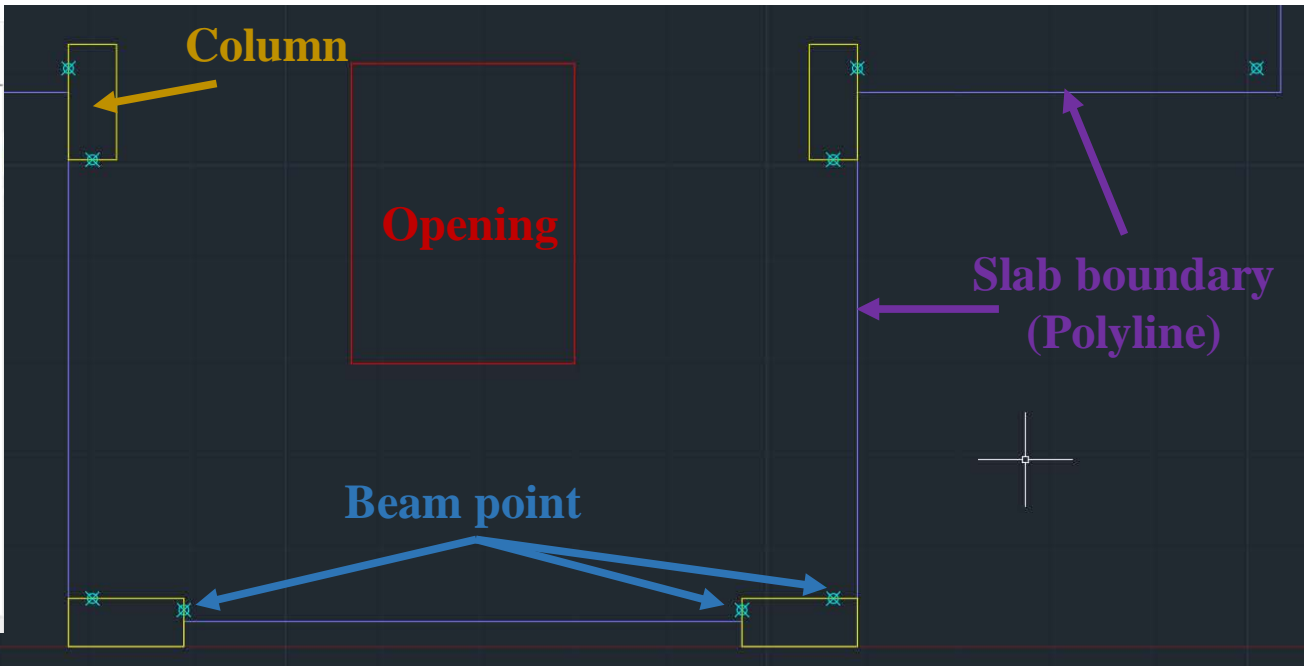
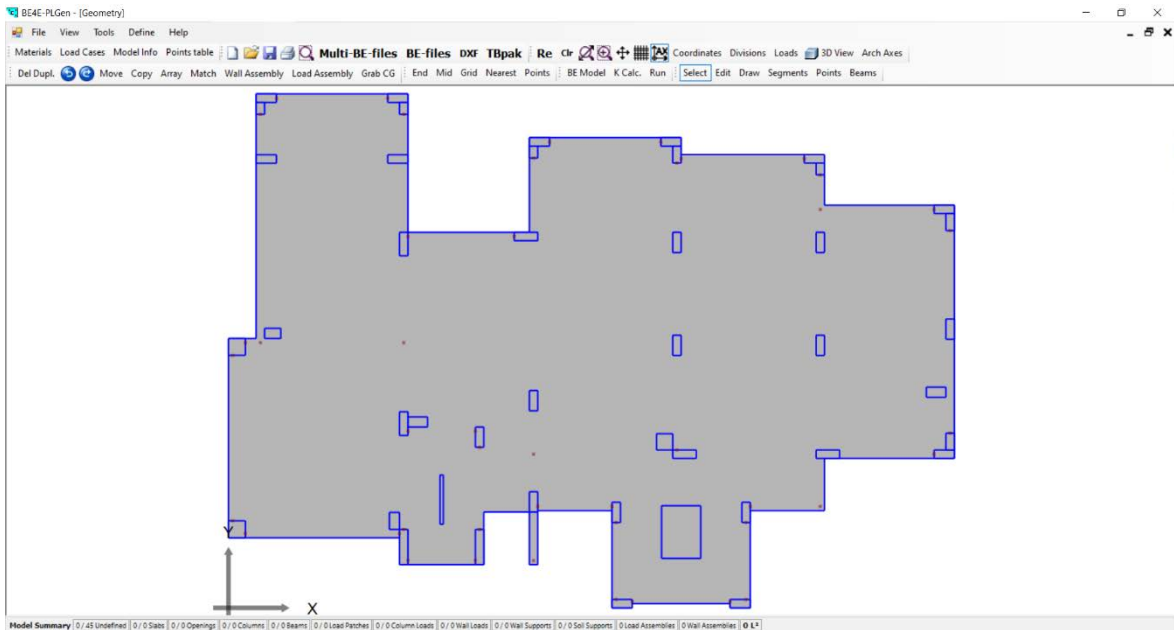
OR



2. Build model and define its elements www.be4e.com

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 as points at start/end of the beams.
- 4- All structural elements are drawn by four points only except the slab and the opening.

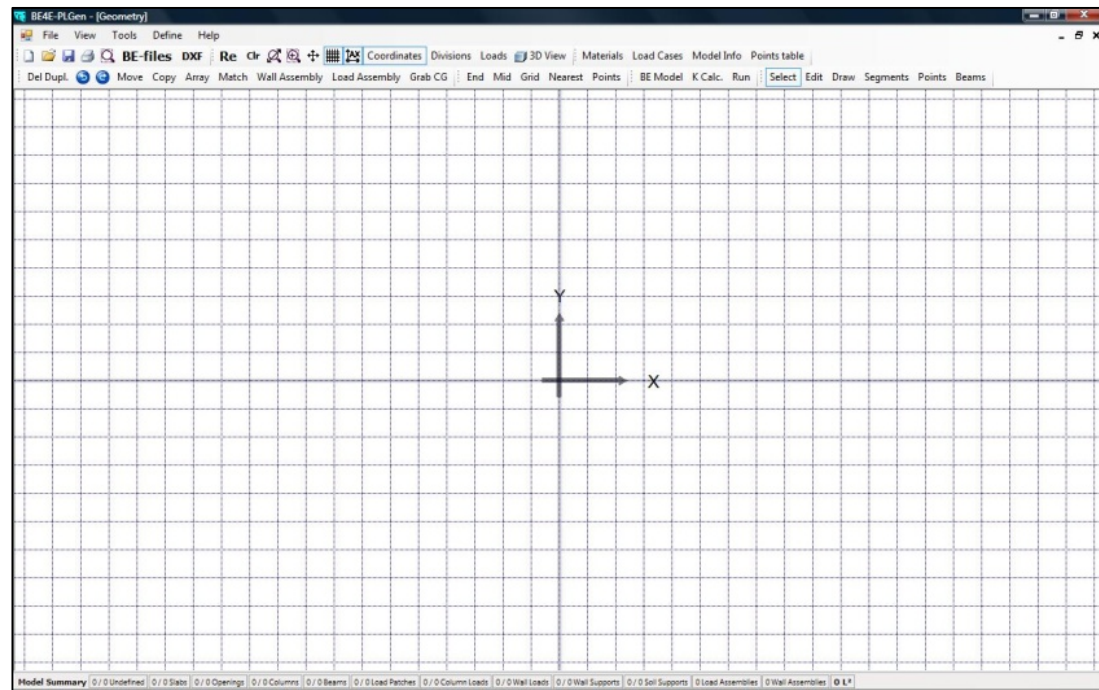


2. Build model and define its elements www.be4e.com

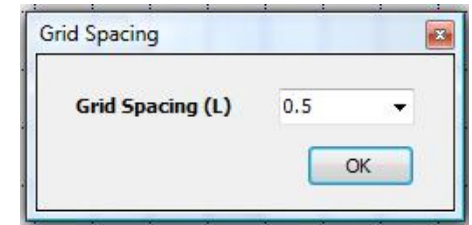
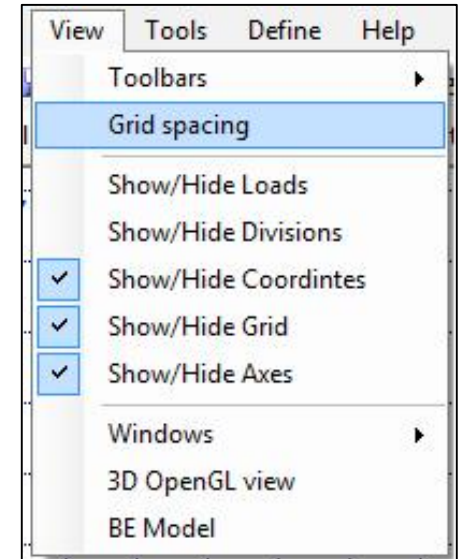
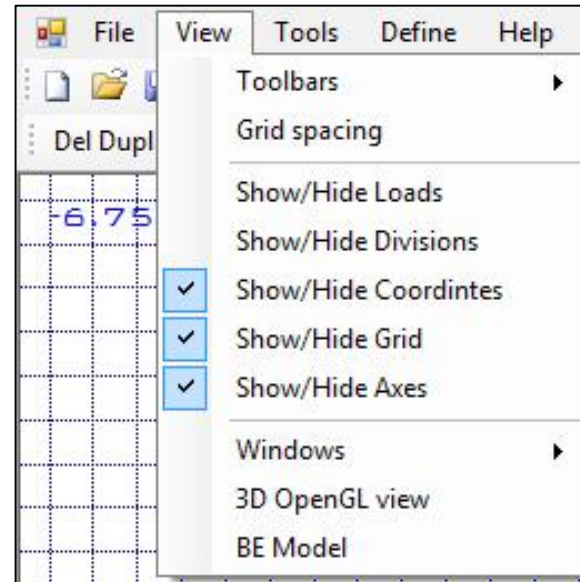
B- Drawing Model from PLGen.

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

The user can change the Grid spacing from view menu.



OR



2. Build model and define its elements www.be4e.com

B- Drawing Model from PLGen.

PLGen has six modes:

Select: to select the object.

Edit: to edit points or geometry of the object.

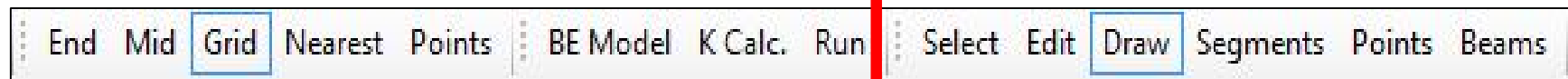
Draw: to draw an object.

Segments: to edit the number of divisions and boundary conditions of the boundary element (openings and slab only).

Points: to draw specific point with specific co-ordinates.

Beams: to draw a beam.

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



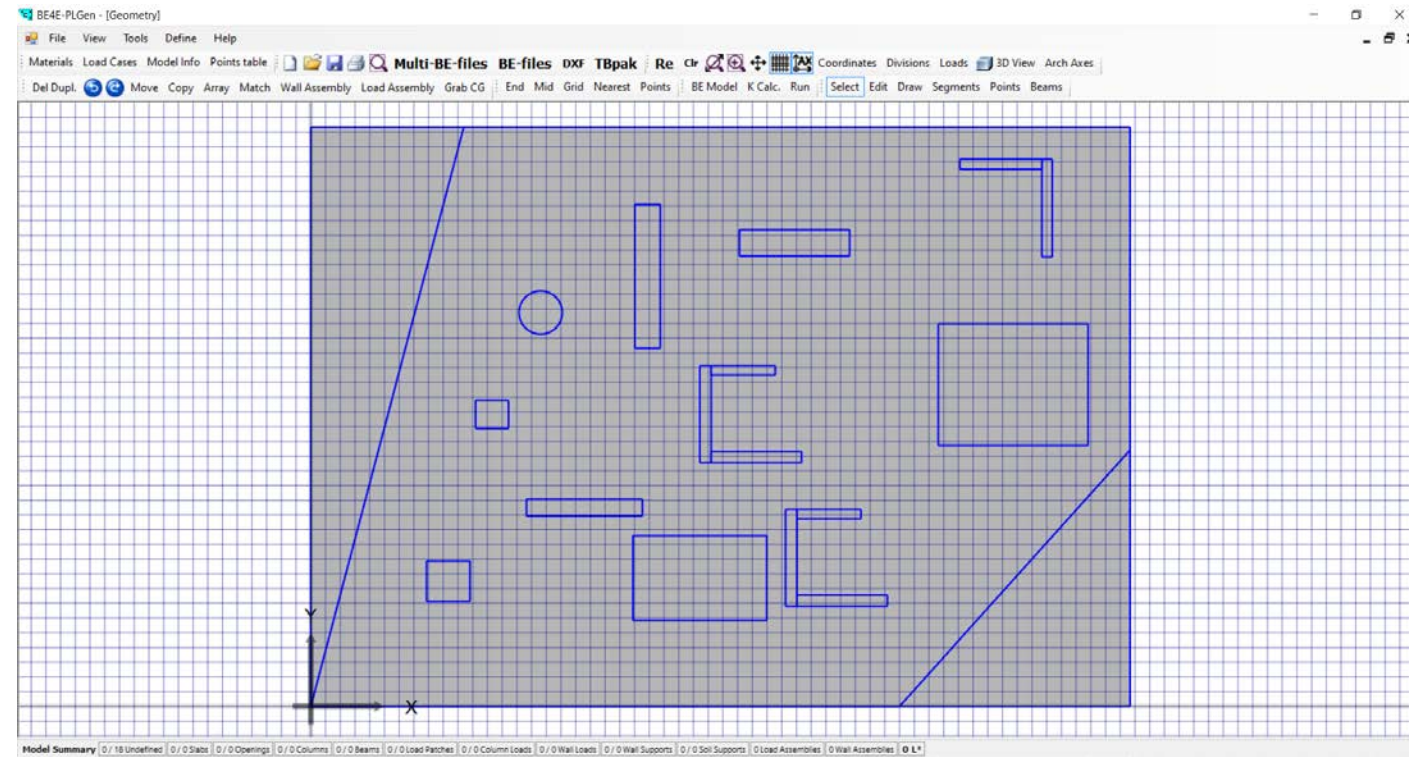
PLGen modes

2. Build model and define its elements www.be4e.com

B- Drawing Model from PLGen.

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
- Then draw the support, load elements, and the opening but still we didn't define them.
- It doesn't matter if we draw clockwise or anticlockwise.
- In order to draw a circle, just click shift button during the drawing process after that press in any place of the outer perimeter of the circle.



2. Build model and define its elements www.be4e.com

Objects in the PLGen can be categorized into three categories:

- Slab and openings (Domain of the problem). → Drawn using any number of points.
- Supporting elements. → Drawn using 4 points only.

Acolumn, Column, Wall support, Awall support, Soil support, Warping wall, Drop.

It has to be noted that, Acolumn, Awall support, Warping wall are special elements used in fixed base package.

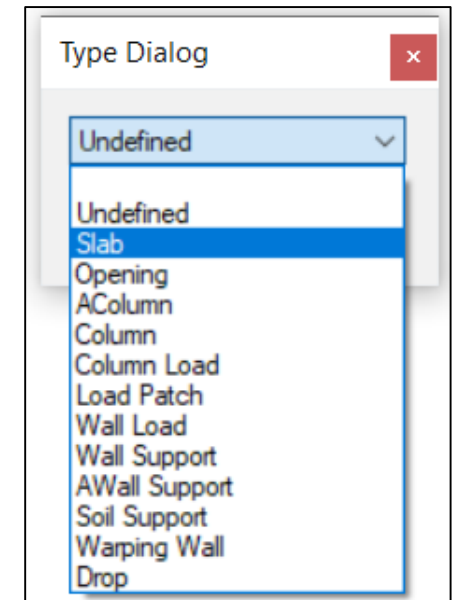
- Loading elements. → Drawn using 4 point only.

Column load, Load patch, Wall load.

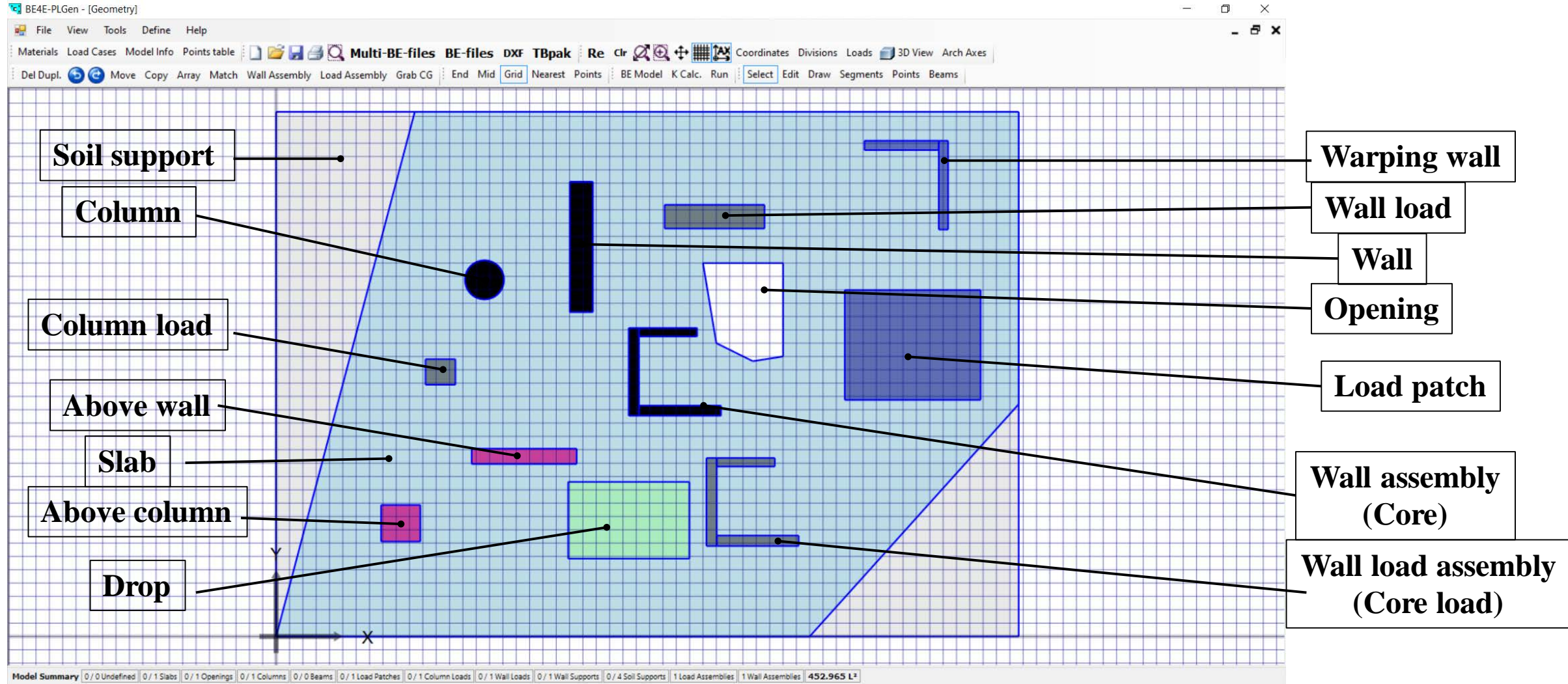
Defining different objects:



- Choose “select mode”.
- Click left click on the object to select it.
- Click right click to choose object type.



2. Build model and define its elements www.be4e.com



The screenshot shows the BE4E-PLGen software interface with a 2D structural model on a grid. The model includes various elements and loads, which are labeled as follows:

- Soil support**: A point on the left boundary.
- Column**: A black circular element.
- Column load**: A small grey rectangular load on a column.
- Above wall**: A pink horizontal rectangular load.
- Slab**: A pink horizontal rectangular load.
- Above column**: A pink square load.
- Drop**: A green square load.
- Warping wall**: A blue L-shaped wall element.
- Wall load**: A grey rectangular load on a wall.
- Wall**: A blue L-shaped wall element.
- Opening**: A white irregular polygonal cutout.
- Load patch**: A blue rectangular load on a wall.
- Wall assembly (Core)**: A blue L-shaped wall element.
- Wall load assembly (Core load)**: A grey rectangular load on a wall.

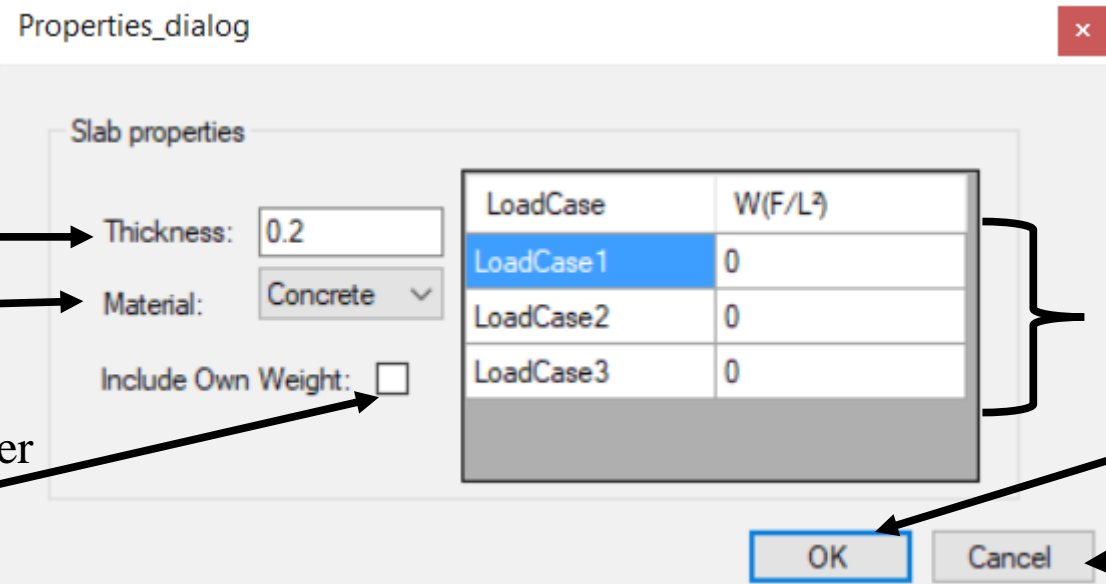
The software interface includes a menu bar (File, View, Tools, Define, Help), a toolbar with various icons, and a status bar at the bottom showing model statistics: 0/0 Undefined, 0/1 Slabs, 0/1 Openings, 0/1 Columns, 0/0 Beams, 0/1 Load Patches, 0/1 Column Loads, 0/1 Wall Loads, 0/1 Wall Supports, 0/4 Soil Supports, 1 Load Assemblies, 1 Wall Assemblies, and a total area of 452.965 L².

2. Build model and define its elements www.be4e.com

Slab

Click on right button on a selected slab to show its properties:

- Define its thickness.
- Define its material.
- Define its domain load for each load case.
- Consider its own weight either by check box or write the weight of slab in the dead load case.



Slab thickness

Slab material

Check this cell to consider the slab own weight

Slab uniform load in each load case

Ok button to save the slab data

Cancel button to dismiss changes in the slab data

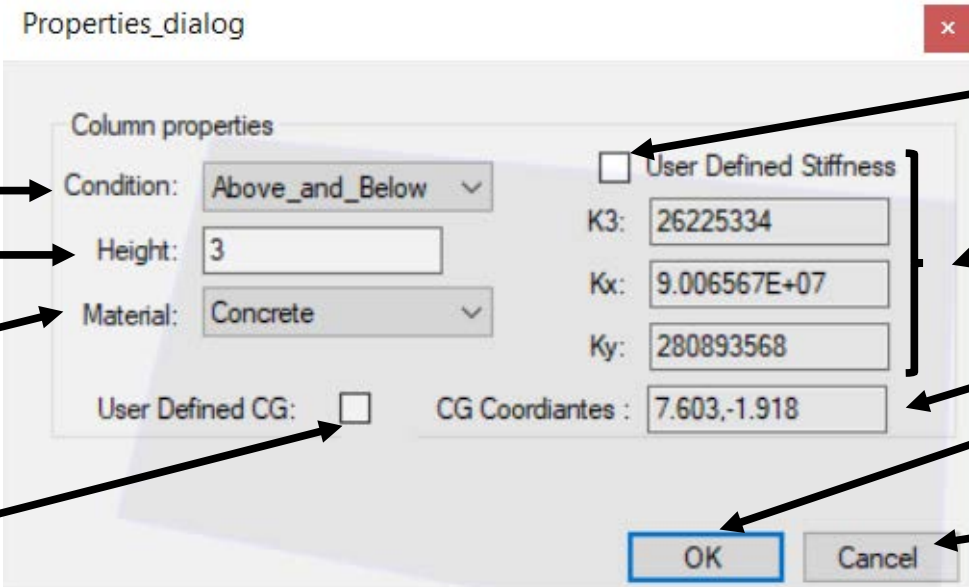
LoadCase	W(F/L ²)
LoadCase1	0
LoadCase2	0
LoadCase3	0

2. Build model and define its elements www.be4e.com

Supporting elements (Column/Above column)

Click on right button on a selected column to show its properties:

- Column contains 3 stiffness (two rotations and one vertical translation DOFs).
- Define its condition (Below only, or above and below the slab).
- Define its height.
- Define its material.
- Column CG and its stiffness are computed automatically, and the user can edit them manually.



Column condition

Column height

Column material

Check this cell to change CG manually

Check this cell to add stiffness manually

Column stiffness

Column CG

Ok button to save the column data

Cancel button to dismiss changes in the column data

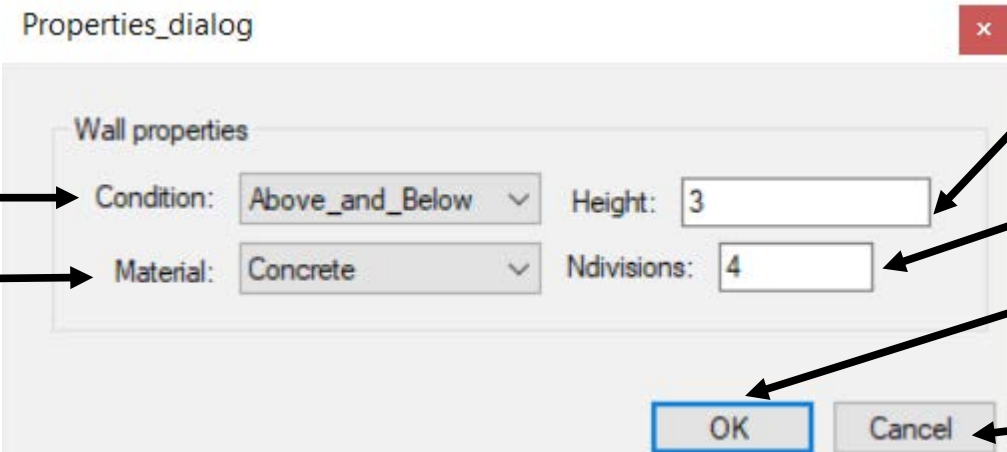
2. Build model and define its elements www.be4e.com

Cairo University

Supporting elements (Wall support/ Above wall support)

Click on right button on a selected wall support to show its properties:

- Wall support is divided into supporting cell, in only one direction, combined together to form wall stiffness.
- Define its condition (Below only, or above and below the slab).
- Define its height.
- Define its material.
- Define its divisions.



Wall condition

Wall material

Wall height

Wall number of divisions

Ok button to save the wall data

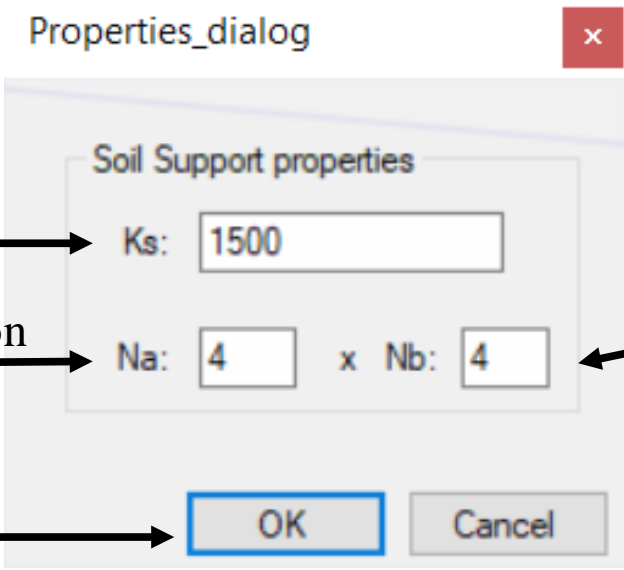
Cancel button to dismiss changes in the wall data

2. Build model and define its elements www.be4e.com

Supporting elements (Soil support)

Click on right button on a selected soil support to show its properties:

- Soil support is divided into supporting cell in two the directions.
- Soil cell contains only vertical translation DOF.
- Define its condition (Below only, or above and below the slab).
- Define its height.
- Define its material.
- Define its divisions.



Soil stiffness / unit area → Ks: 1500

Soil number of divisions in 'a' direction → Na: 4 x Nb: 4

Soil number of divisions in 'b' direction → Nb: 4

Ok button to save the soil data → OK

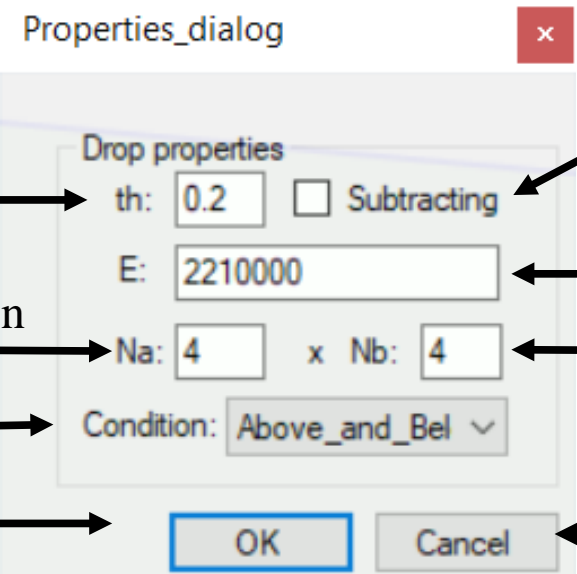
Cancel button to dismiss changes in the soil data → Cancel

2. Build model and define its elements www.be4e.com

Supporting elements (Drop)

Click on right button on a selected drop to show its properties:

- Drop is divided into supporting cell in two the directions.
- Drop cell contains three DOFs (two rotations and one vertical translation).
- Define its thickness (The additional or subtracting thickness without slab thickness).
- Define its modulus of elasticity.
- Define its condition (Above and below, below only, or as assigned).
- Define its divisions.



Drop thickness → th: 0.2

Drop number of divisions in 'a' direction → Na: 4

Drop condition → Condition: Above_and_Bel

Ok button to save the drop data → OK

Cancel button to dismiss changes in the drop data → Cancel

Check box incase of drop thickness is subtracted from slab thickness not added → Subtracting

Drop modulus of elasticity → E: 2210000

Drop number of divisions in 'b' direction → Nb: 4

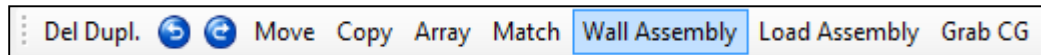
2. Build model and define its elements www.be4e.com

Supporting elements (Wall assembly)

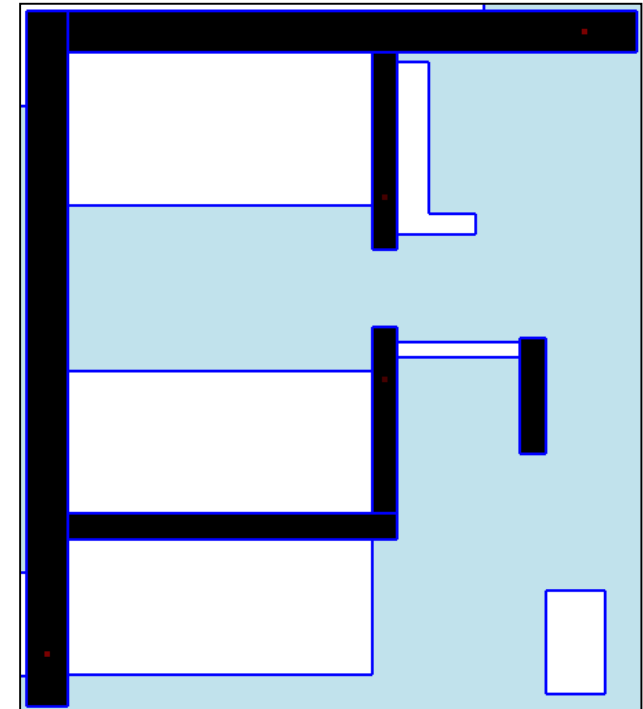
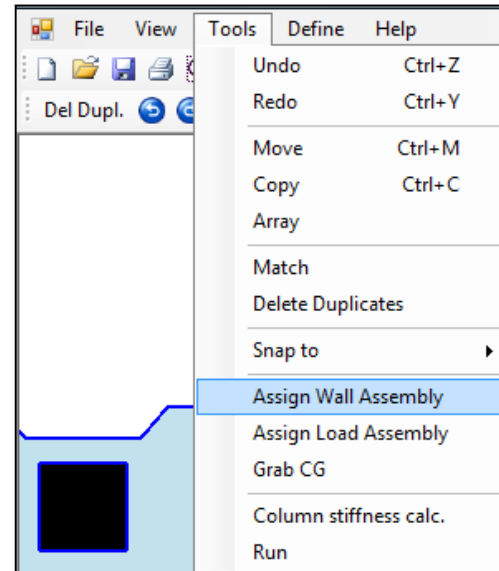
Assemblies is used if the user want to combine either support together like cores or any shape of shear walls etc.

- 1- Assign wall assembly (core) elements as wall support.
- 2- Select all walls that need to be combined then click on wall assembly button.

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.



OR

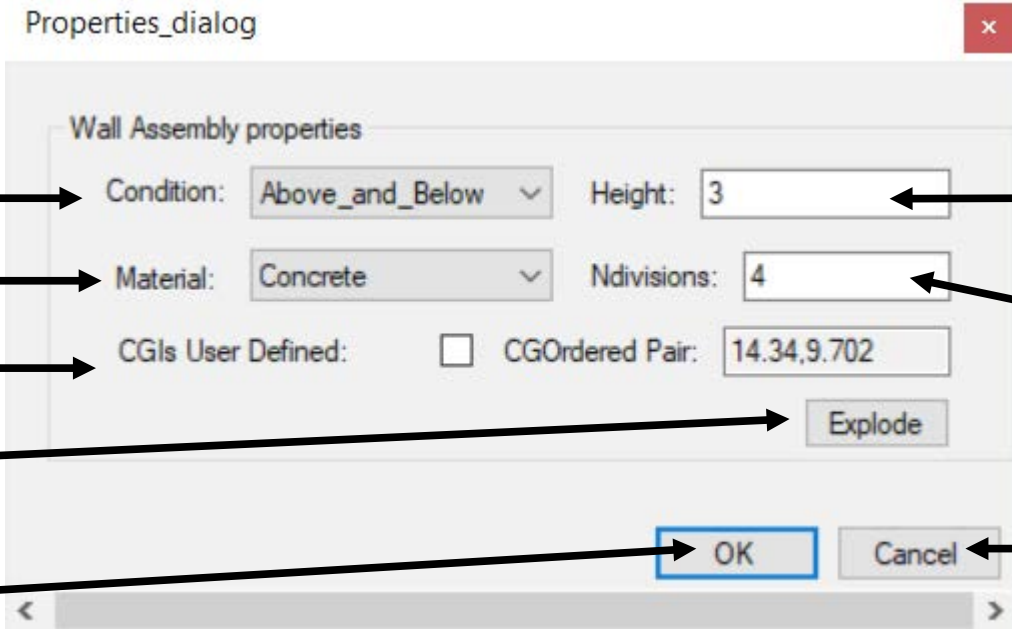


2. Build model and define its elements www.be4e.com

Supporting elements (Wall assembly)

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

The Explode tab restore the five shear walls again.



Wall assembly condition → Condition: Above_and_Below

Wall assembly material → Material: Concrete

Wall assembly CG → CGIs User Defined: CGOrdered Pair: 14.34,9.702

Explode tab to restore the shear walls again → Explode

Ok button to save the wall assembly data → OK

Wall assembly height → Height: 3

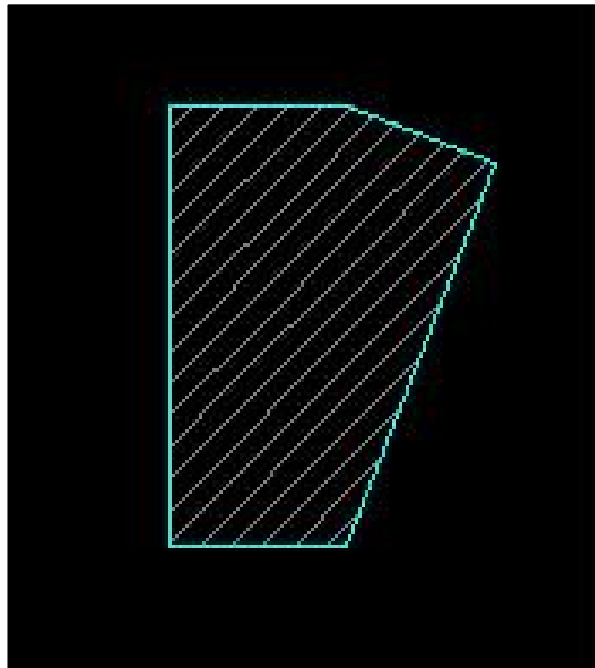
Wall assembly number of division per each wall → Ndivisions: 4

Cancel button to dismiss changes in the wall assembly data → Cancel

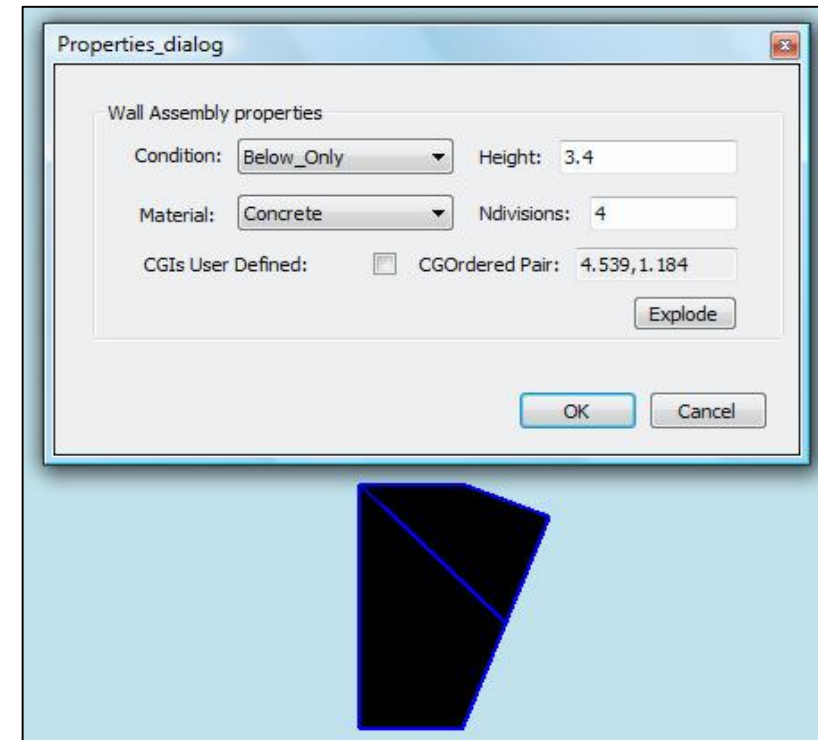
2. Build model and define its elements www.be4e.com

Supporting elements (Non-quadratic column)

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



Non-quadratic column

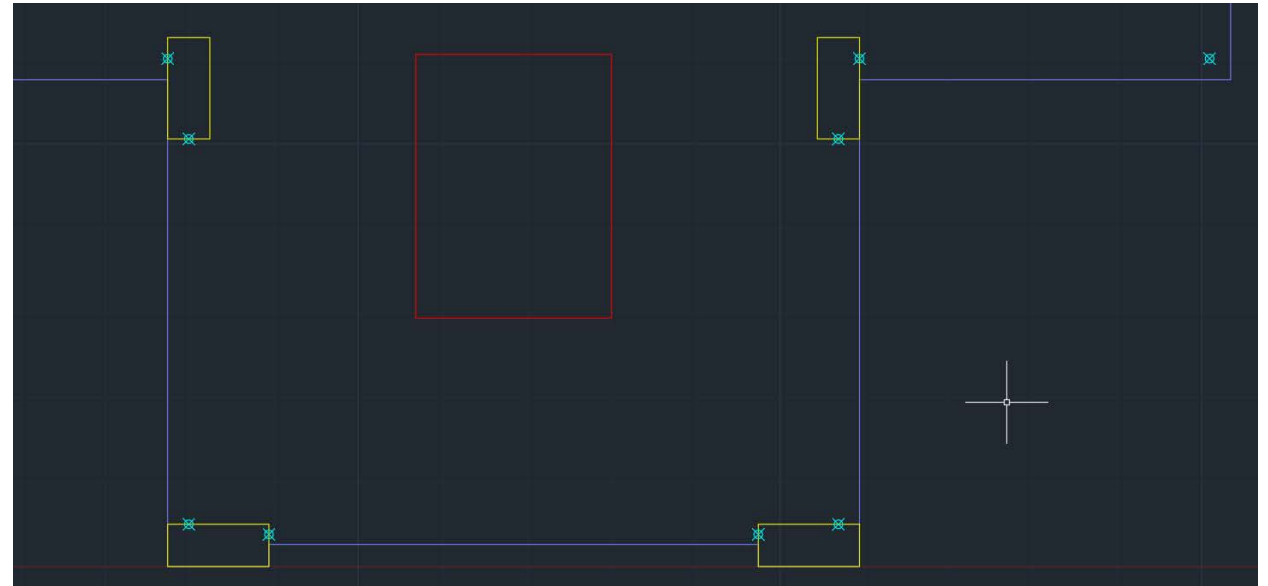
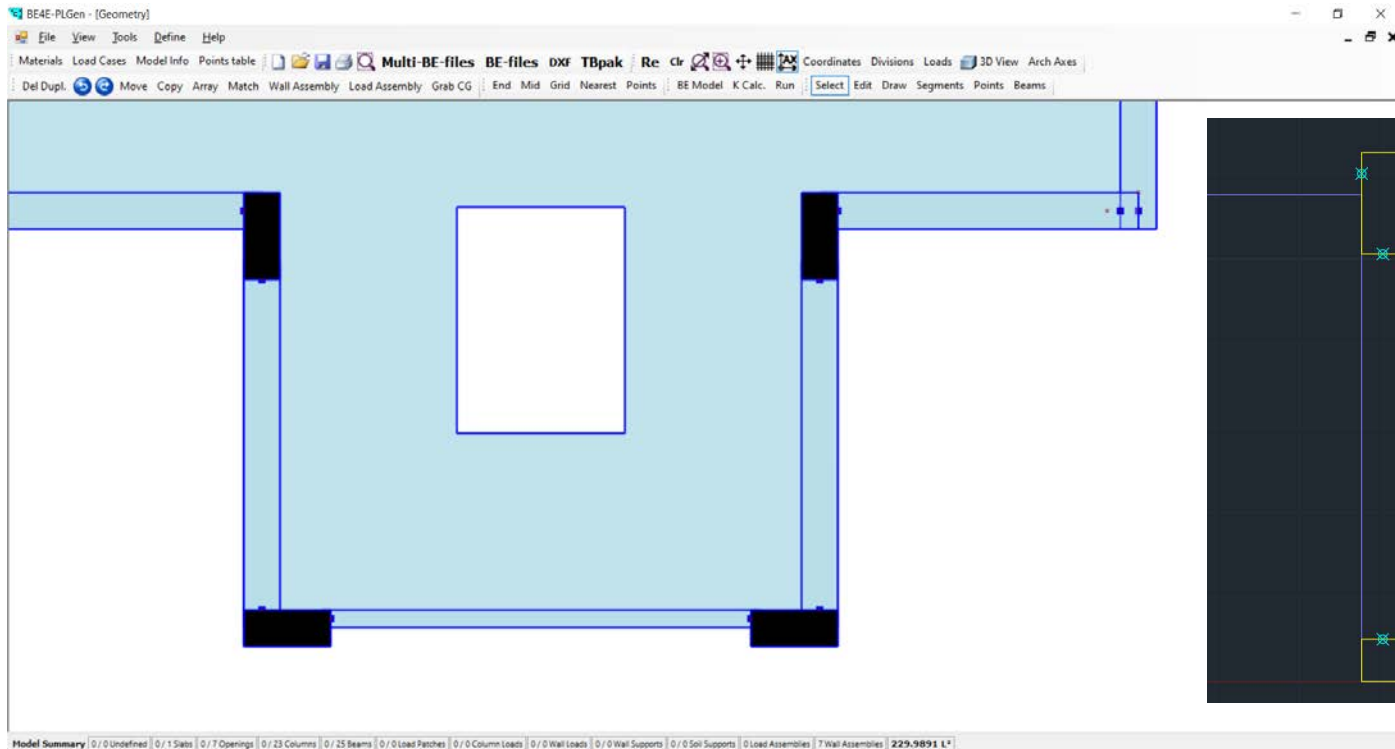
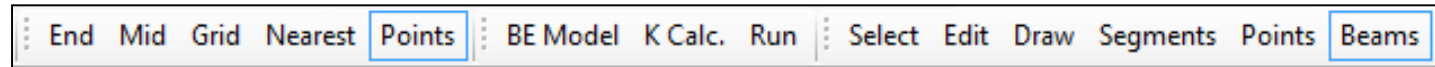


Non-quadratic column in PLGen

2. Build model and define its elements www.be4e.com

Supporting elements (Beams)

- First change PLGen mode to Beams.
- Beams can be modeled by drawing them from PLGen using sniping tools, or by importing the beam start/end as points from AutoCAD (DXF) file.

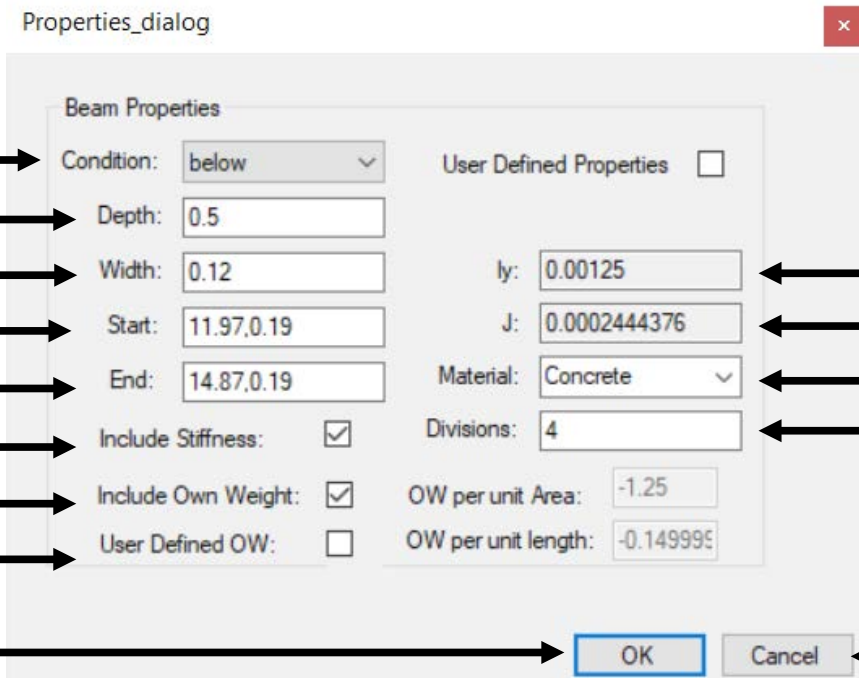


2. Build model and define its elements www.be4e.com

Supporting elements (Beams)

For showing beam properties the user should change from beam mode to select mode, then click on right button on a selected beam to show its properties:

- 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).



The screenshot shows the 'Properties_dialog' window with the following fields and controls:

- Condition:** dropdown menu set to 'below'
- Depth:** text input field with '0.5'
- Width:** text input field with '0.12'
- Start:** text input field with '11.97,0.19'
- End:** text input field with '14.87,0.19'
- Include Stiffness:** checked checkbox
- Include Own Weight:** checked checkbox
- User Defined OW:** unchecked checkbox
- User Defined Properties:** unchecked checkbox
- ly:** text input field with '0.00125'
- J:** text input field with '0.0002444376'
- Material:** dropdown menu set to 'Concrete'
- Divisions:** text input field with '4'
- OW per unit Area:** text input field with '-1.25'
- OW per unit length:** text input field with '-0.149999'
- Buttons:** 'OK' and 'Cancel' buttons at the bottom right.

Annotations with arrows point to the following elements:

- Beam condition
- Beam depth
- Beam width
- Beam start point coordinates
- Beam end point coordinates
- Beam stiffness
- Beam OW check box
- User defined beam OW check box
- Ok button to save the beam data
- Beam inertia
- Beam torsion constant
- Beam material
- Beam divisions
- Cancel button to dismiss changes in the beam data

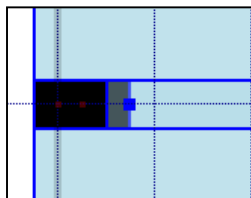
2. Build model and define its elements www.be4e.com

Supporting elements (Beams)

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

In case of hinged beams

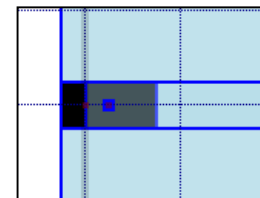
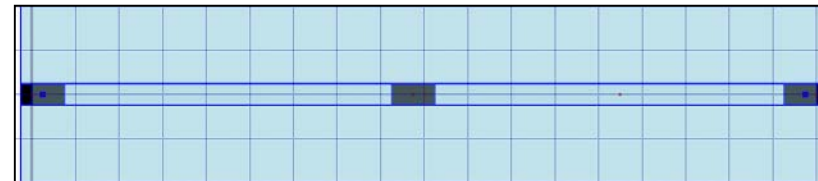
The beam inserted in column less than or equal to the half beam width.



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

In case of fixed beams

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



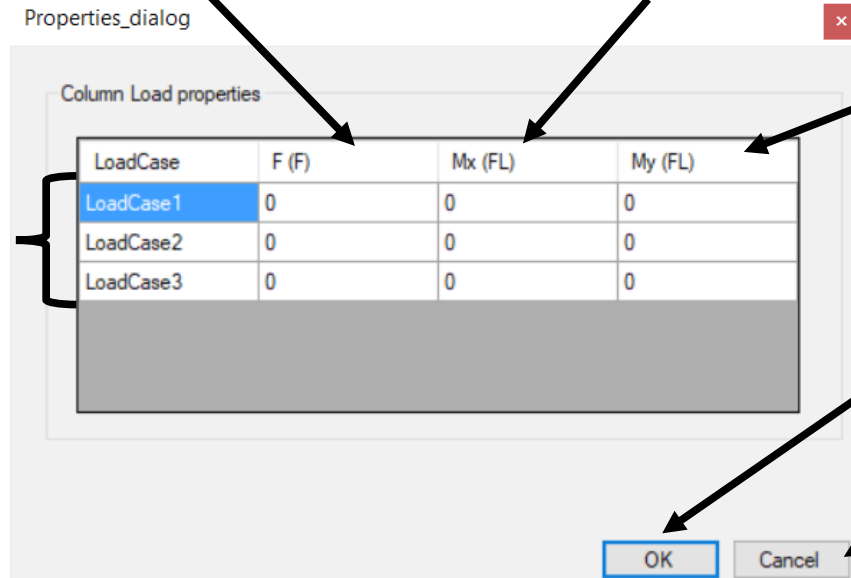
$$L_{in} \geq 0.5 B_{width}$$

2. Build model and define its elements www.be4e.com

Loading elements (Column load)

Click on right button on a selected column load to show its properties:

- Column contains three load values (two moments and one vertical force).
- Define column loads for each load case.



Column vertical force

Column bending moment about x-axis (M_{xx})

Column bending moment about y-axis (M_{yy})

Load cases

LoadCase	F (F)	Mx (FL)	My (FL)
LoadCase1	0	0	0
LoadCase2	0	0	0
LoadCase3	0	0	0

Ok button to save the column data

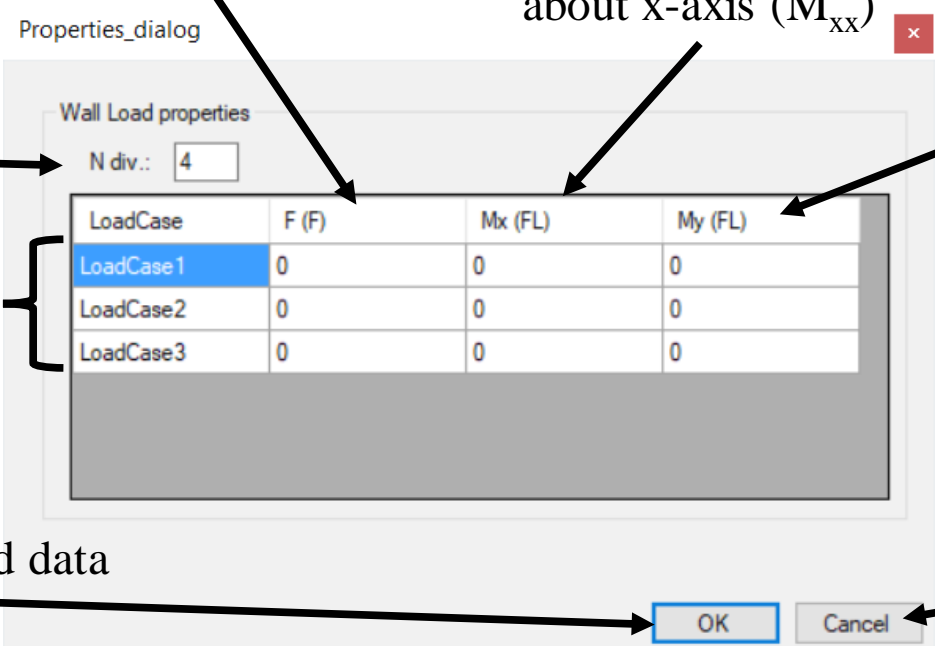
Cancel button to dismiss changes in the column data

2. Build model and define its elements www.be4e.com

Loading elements (Wall load)

Click on right button on a selected wall load to show its properties:

- Wall load is divided into loading cell, in only one direction, combined together to form wall load.
- Define wall loads for each load case.



Wall load vertical force

Wall load bending moment about x-axis (M_{xx})

Wall load bending moment about y-axis (M_{yy})

Wall load number of divisions

Load cases

LoadCase	F (F)	Mx (FL)	My (FL)
LoadCase1	0	0	0
LoadCase2	0	0	0
LoadCase3	0	0	0

Ok button to save the wall load data

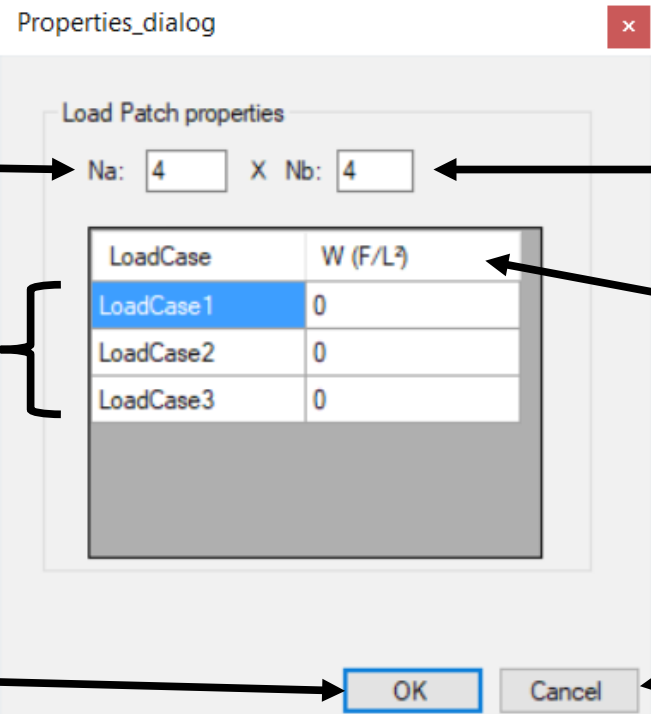
Cancel button to dismiss changes in the wall load data

2. Build model and define its elements www.be4e.com

Loading elements (Load patch)

Click on right button on a selected load patch to show its properties:

- Load patch is divided into loading cells in two the directions.
- load cell contains only vertical uniform load.
- Define its uniform load for each load case.



The screenshot shows the 'Load Patch properties' dialog box with the following fields and table:

Na: X Nb:

LoadCase	W (F/L ²)
LoadCase1	0
LoadCase2	0
LoadCase3	0

Buttons: OK, Cancel

Annotations:

- Load patch number of divisions in 'a' direction (points to Na)
- Load patch number of divisions in 'b' direction (points to Nb)
- Load cases (bracketed next to the table)
- Uniform load value (points to the 'W (F/L²)' column)
- Ok button to save the load patch data (points to OK)
- Cancel button to dismiss changes in the load patch data (points to Cancel)

2. Build model and define its elements www.be4e.com

Loading elements (Wall load assembly)

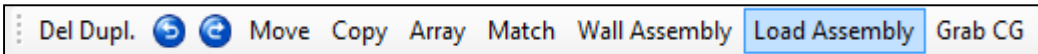
Load assemblies is used if the user want to combine either load together like core's load or any shape of shear wall's load.

1- Assign wall load assembly (core) elements as wall load.

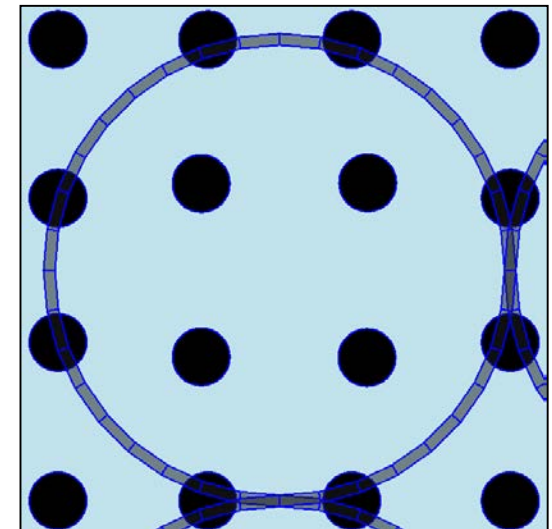
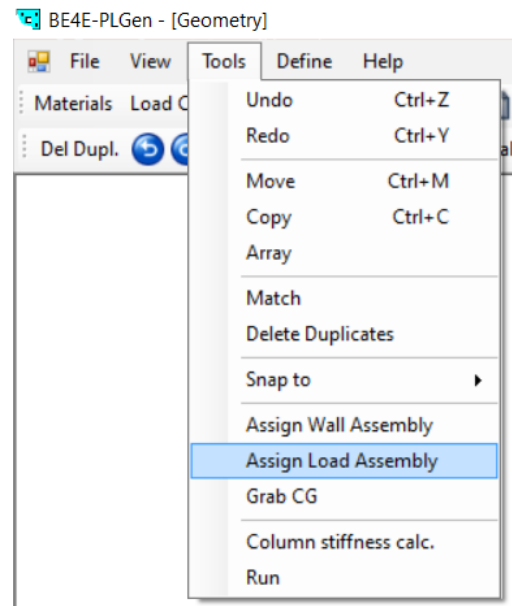
2- Select all wall loads that need to be combined then click on wall assembly button.

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

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



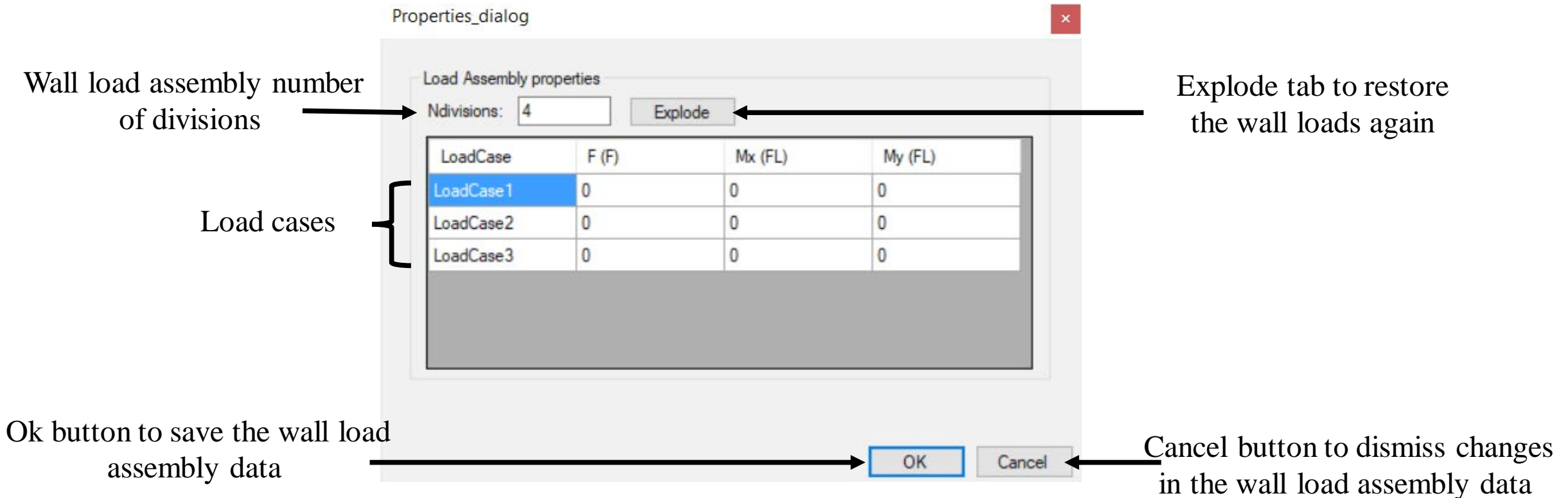
OR



2. Build model and define its elements www.be4e.com

Loading elements (Wall load assembly)

- Define wall load assembly for each load case.
- The explode tab restore the wall loads again.



Wall load assembly number of divisions

Load cases

Explode tab to restore the wall loads again

Ok button to save the wall load assembly data

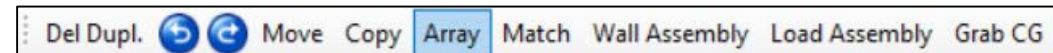
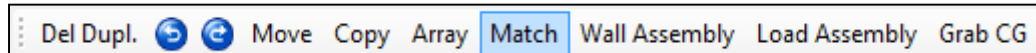
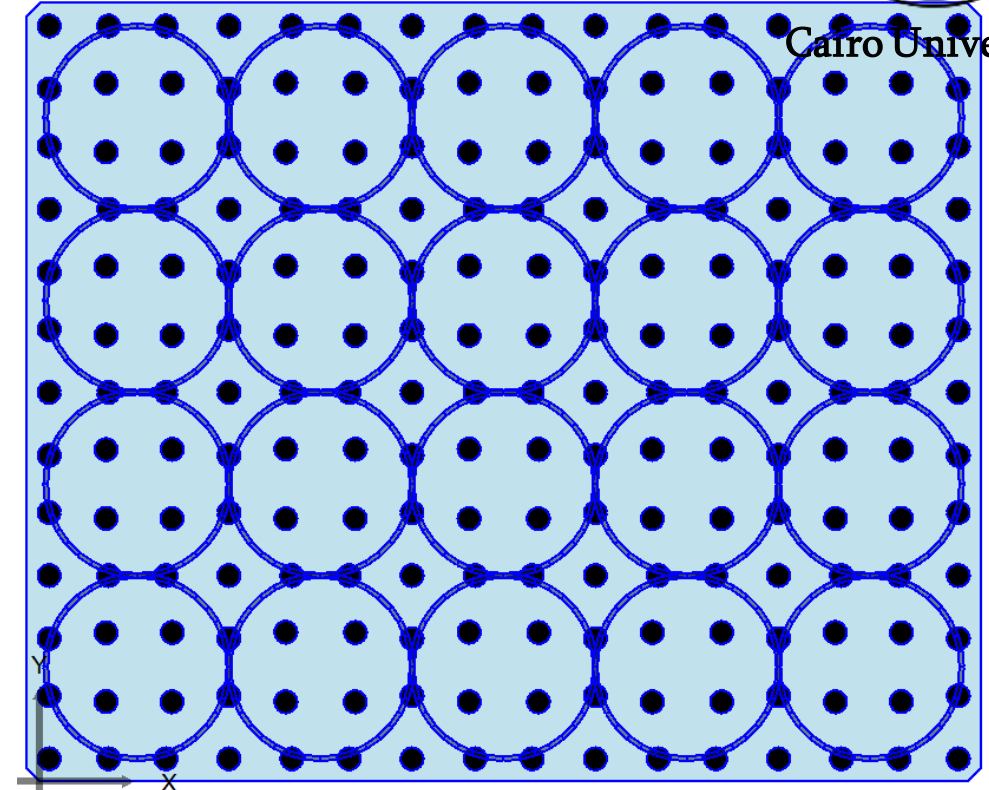
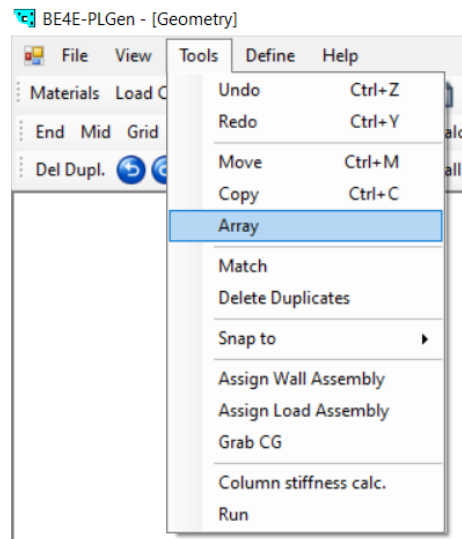
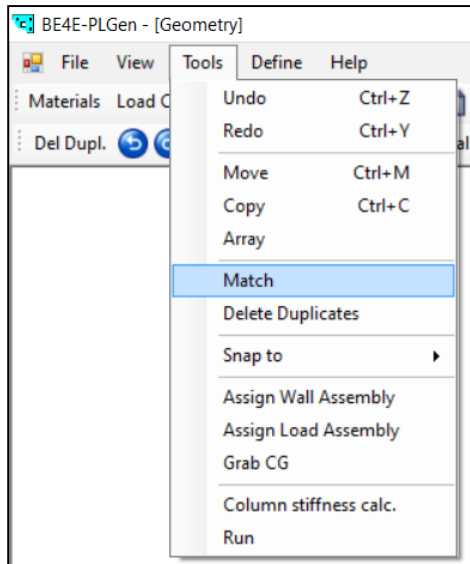
Cancel button to dismiss changes in the wall load assembly data

LoadCase	F (F)	Mx (FL)	My (FL)
LoadCase1	0	0	0
LoadCase2	0	0	0
LoadCase3	0	0	0

2. Build model and define its elements www.be4e.com

Other PLGen tools

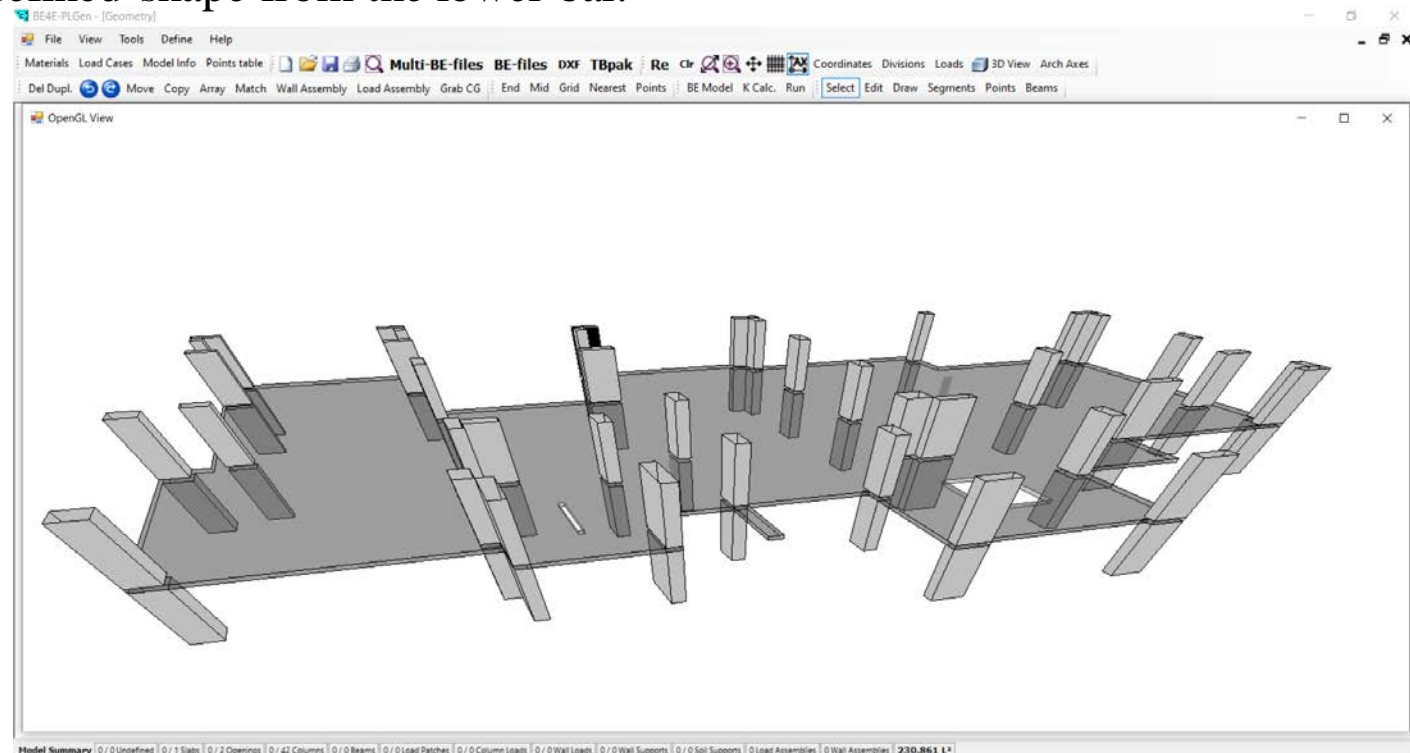
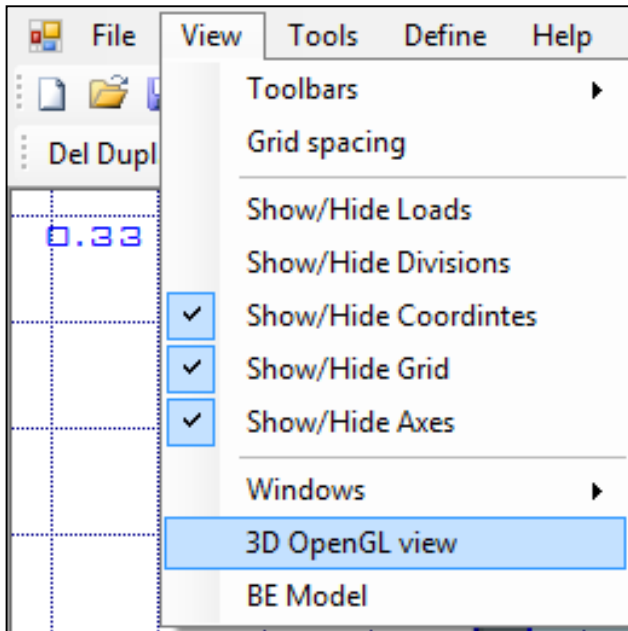
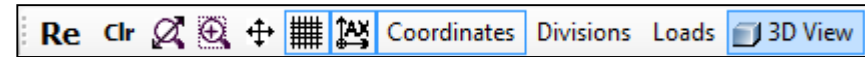
- Array tool to draw array of an element.
- Match tool to match prosperities from element to another ones.



2. Build model and define its elements www.be4e.com

Other PLGen tools

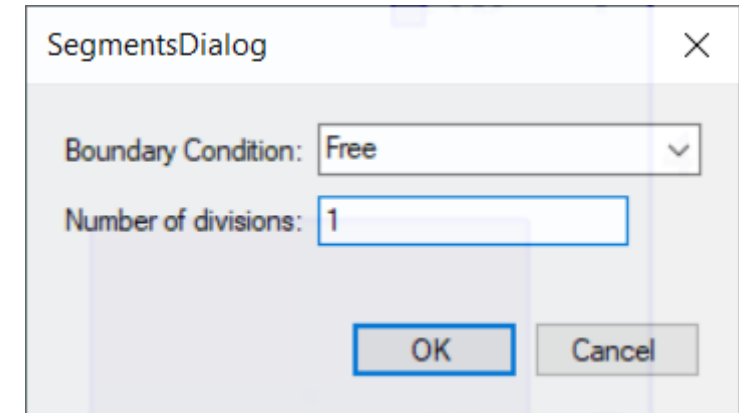
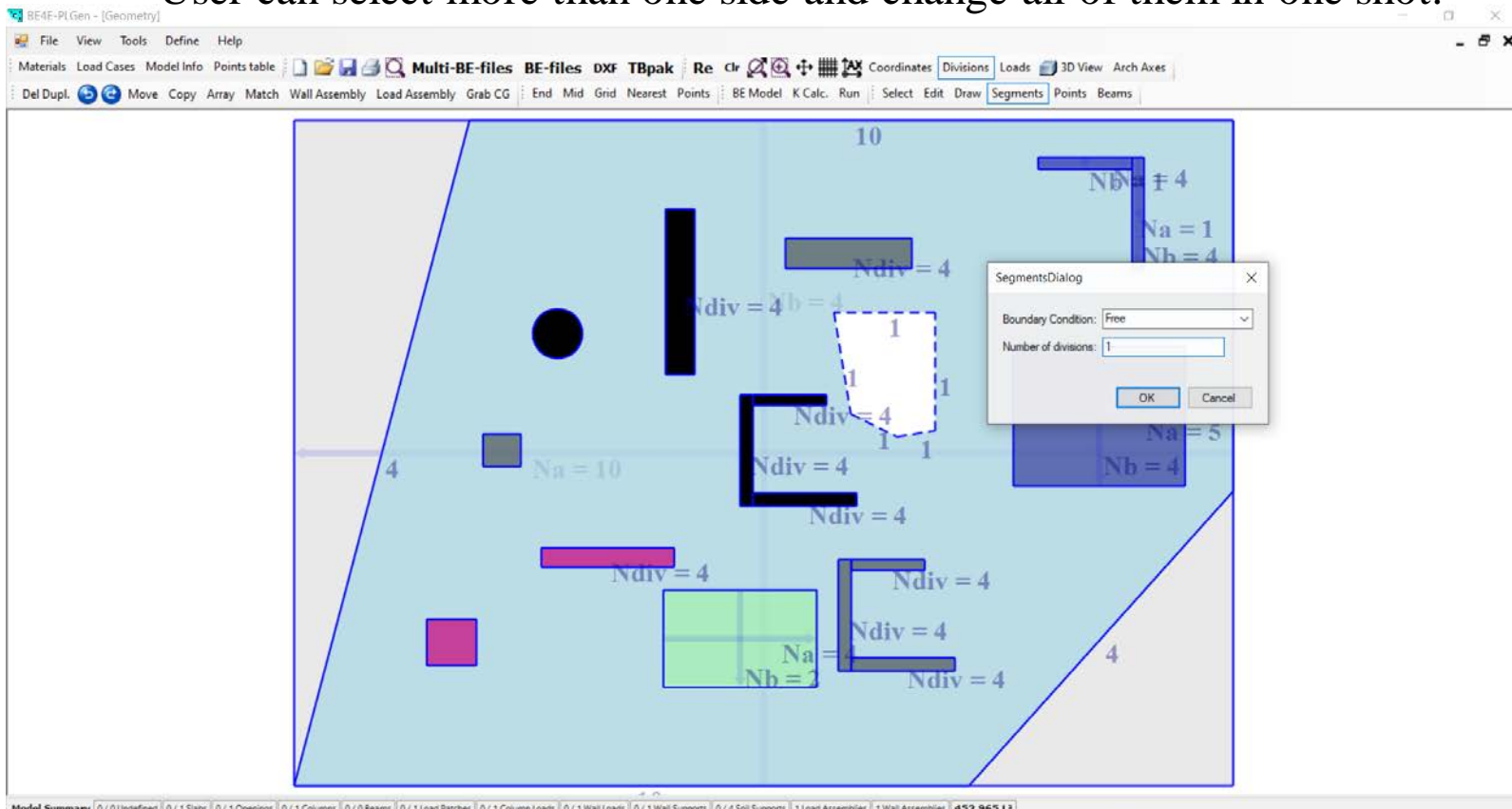
- 3D view of the model after finishing the generator file.
- The user can also check that there is not any undefined shape from the lower bar.



3. Edit the boundary element division

www.be4e.com

- The default number of boundary elements is four.
- Change PLGen mode to Segments mode.
- 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 / boundary elements.
- User can select more than one side and change all of them in one shot.



A. PLGen – Model generator

B. PLView – Numerical model

C. PLCoreMan – Manager and solver

D. PLPost – Post processing

E. PLDesign – Design tool

F. PLPAK modelling capabilities

1. Edit model information ✓

2. Build model and define its elements ✓

3. Edit the boundary element divisions ✓

CUFE-BE



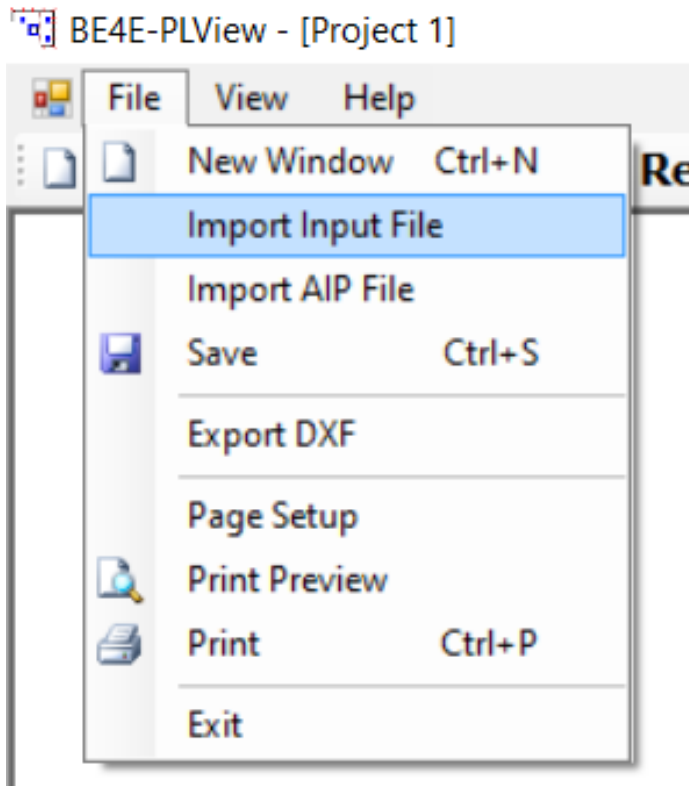
www.be4e.com



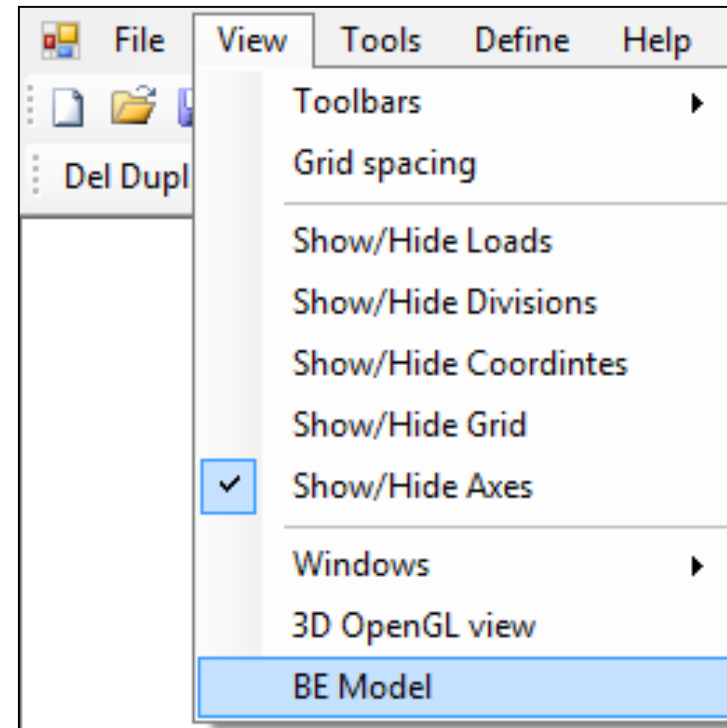
Cairo University

- A. PLGen – Model generator
- B. PLView – Numerical model**
- C. PLCoreMan – Manager and solver
- D. PLPost – Post processing
- E. PLDesign – Design tool
- F. PLPAK modelling capabilities

- PLView is used to view and check the boundary element numerical model before running the model.
- User can check all entered information.
- User can open PLView directly then load the *.in file, or from PLGen by clicking BE Model button.

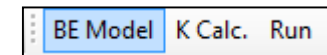


From PLView



From PLGen

OR



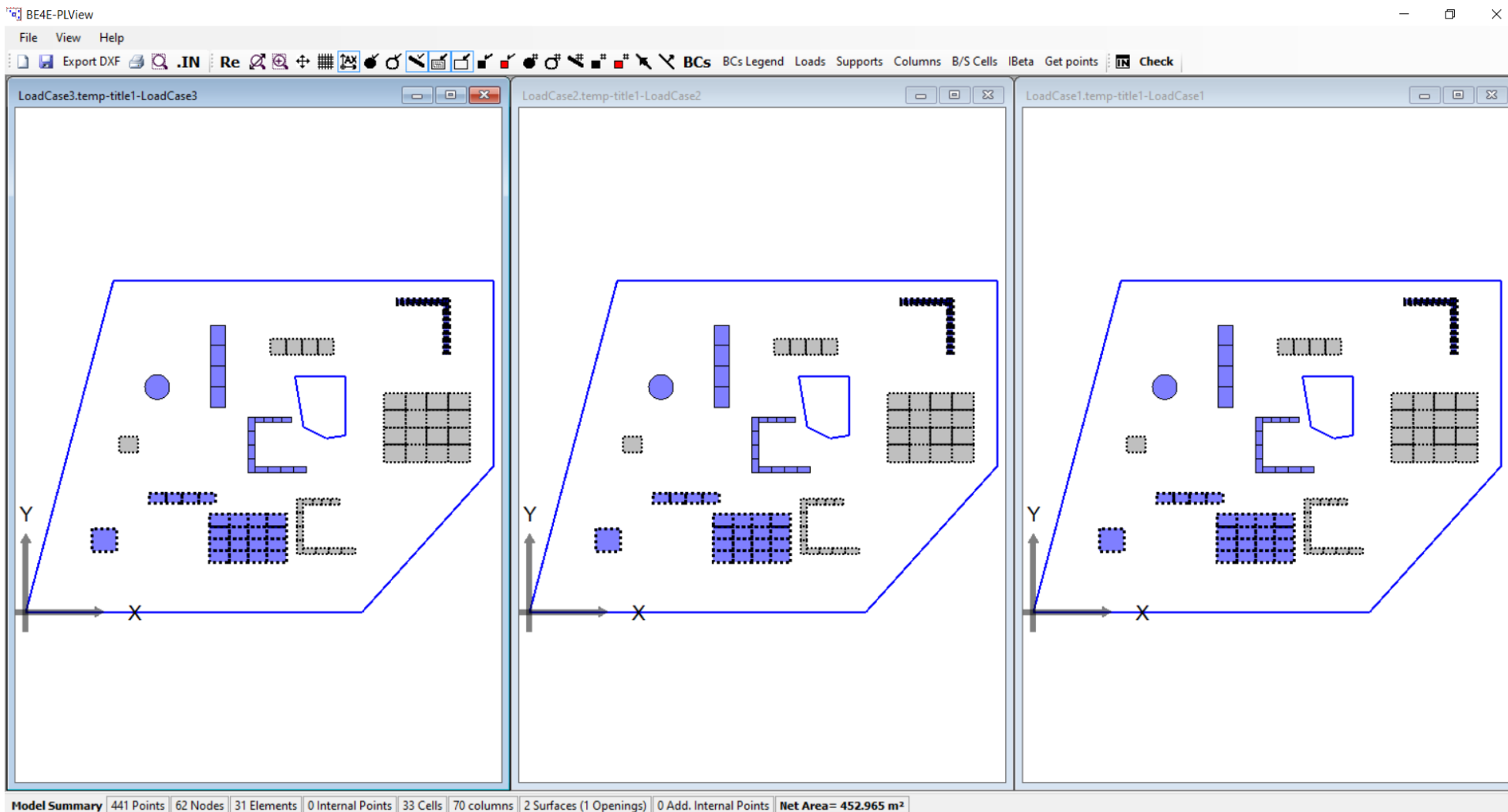
CUFE-BE



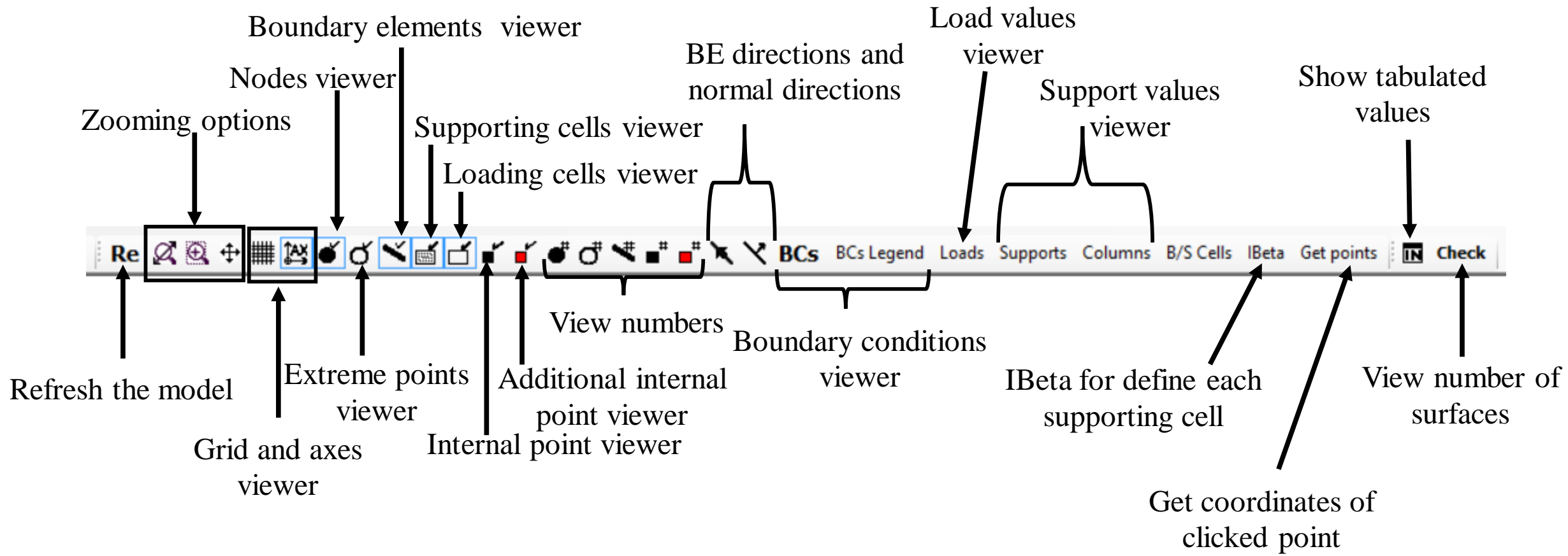
- PLView open window for each load case.
- The model will be unable to run in cases of, two node have the same coordinates or two supporting cells have the same center.

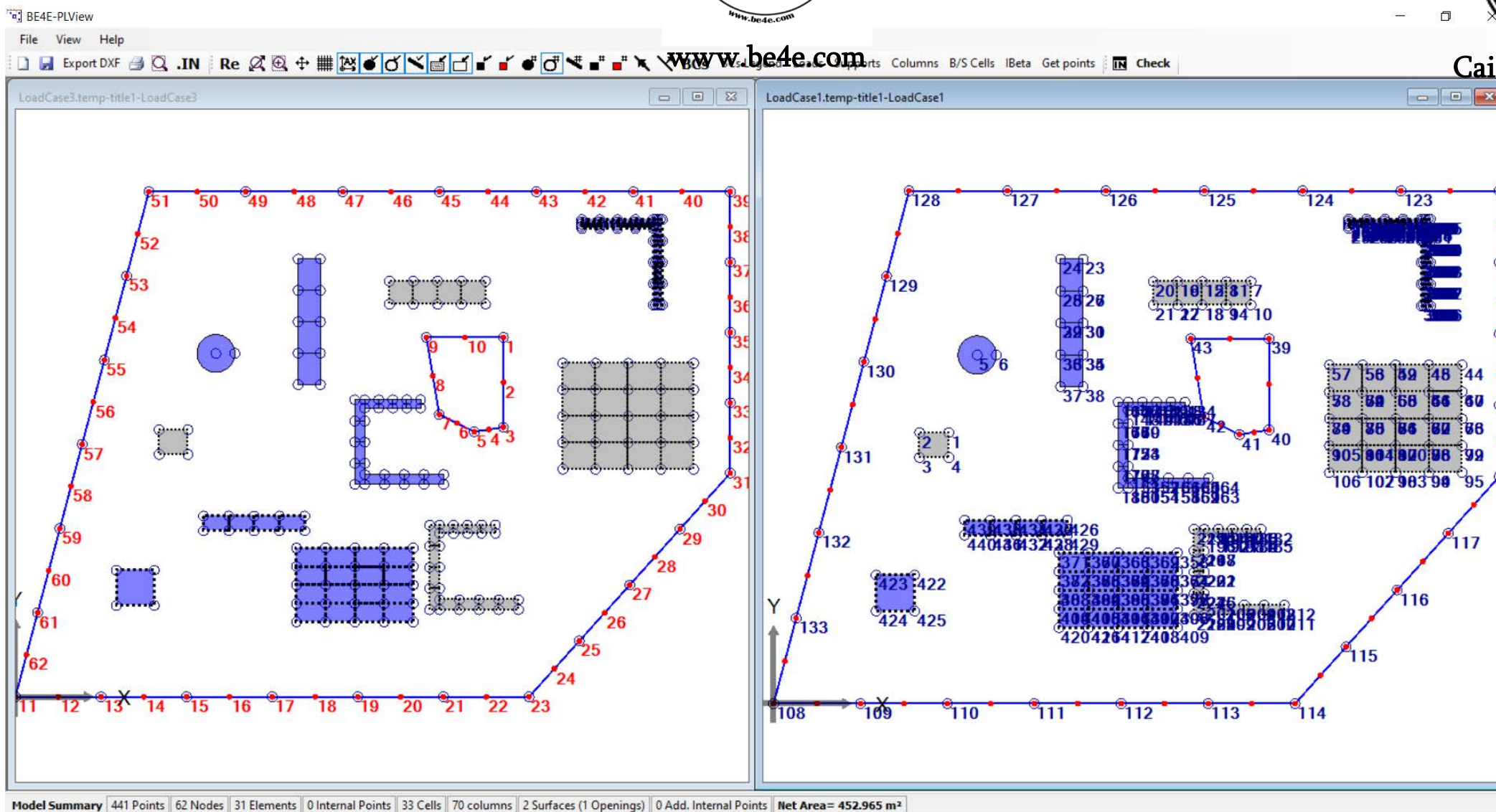
www.be4e.com

Cairo University

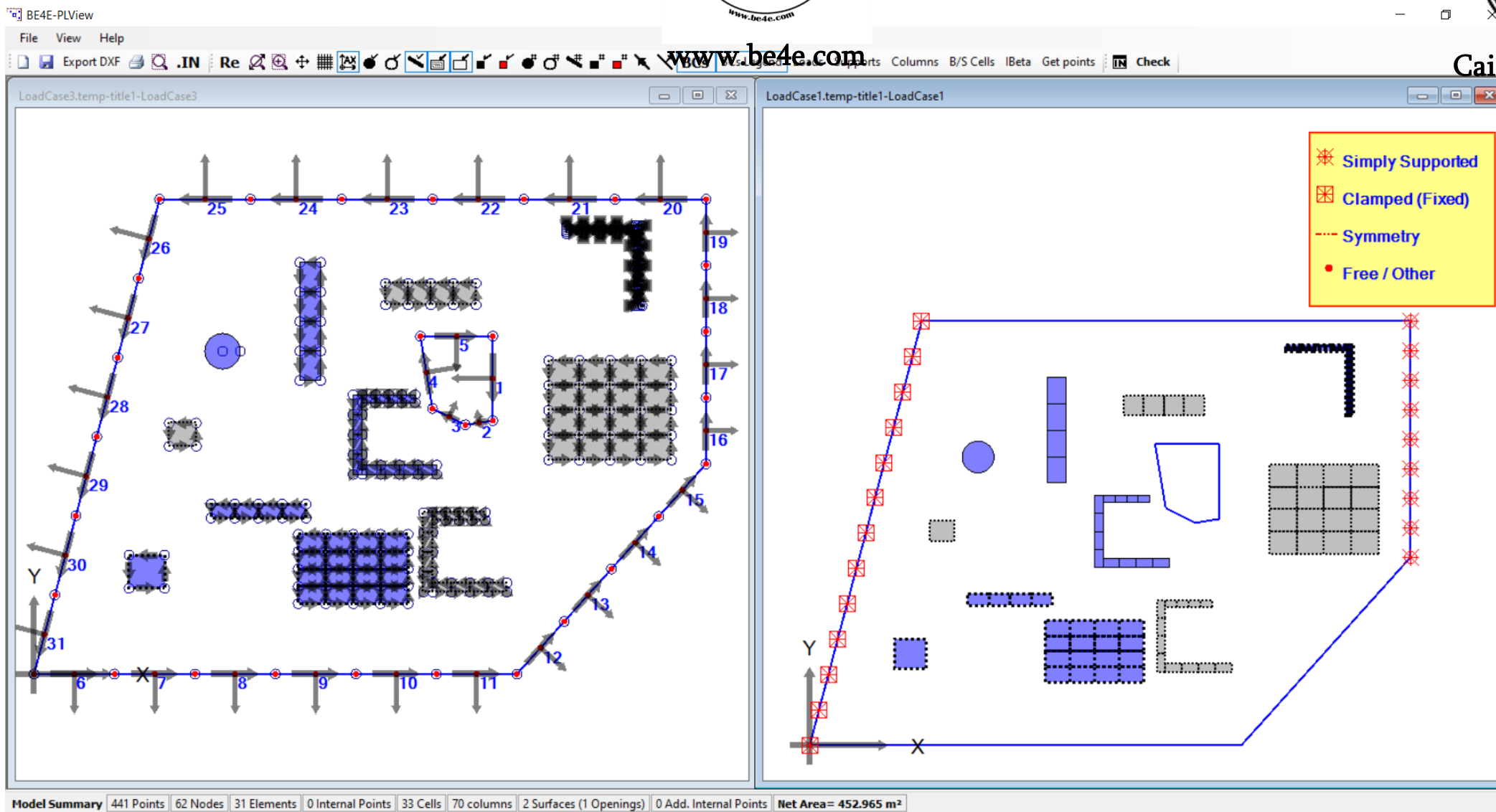


- In PLView user can use these features.



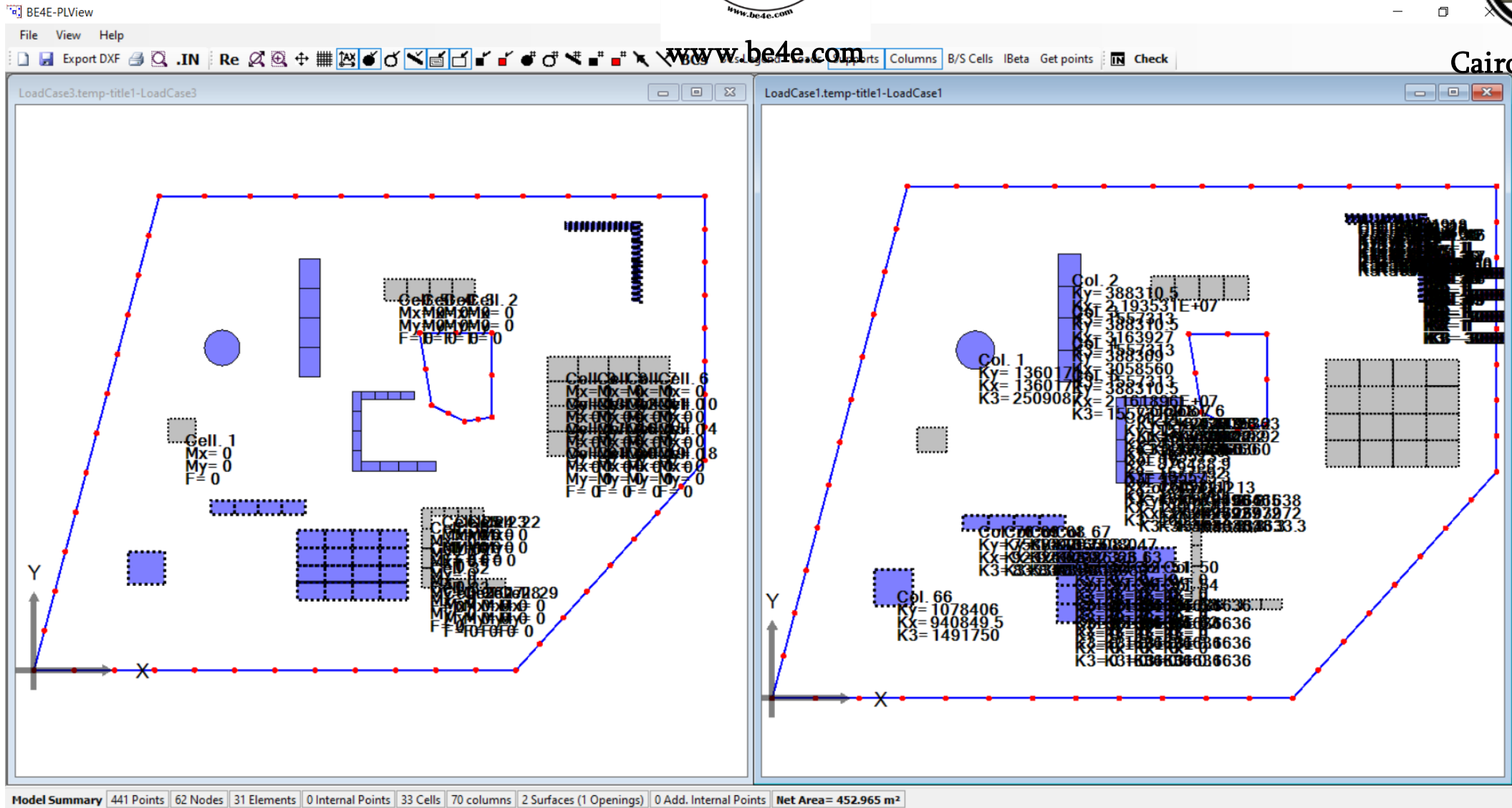


BE model of slab showing nodes, and extreme points numbers



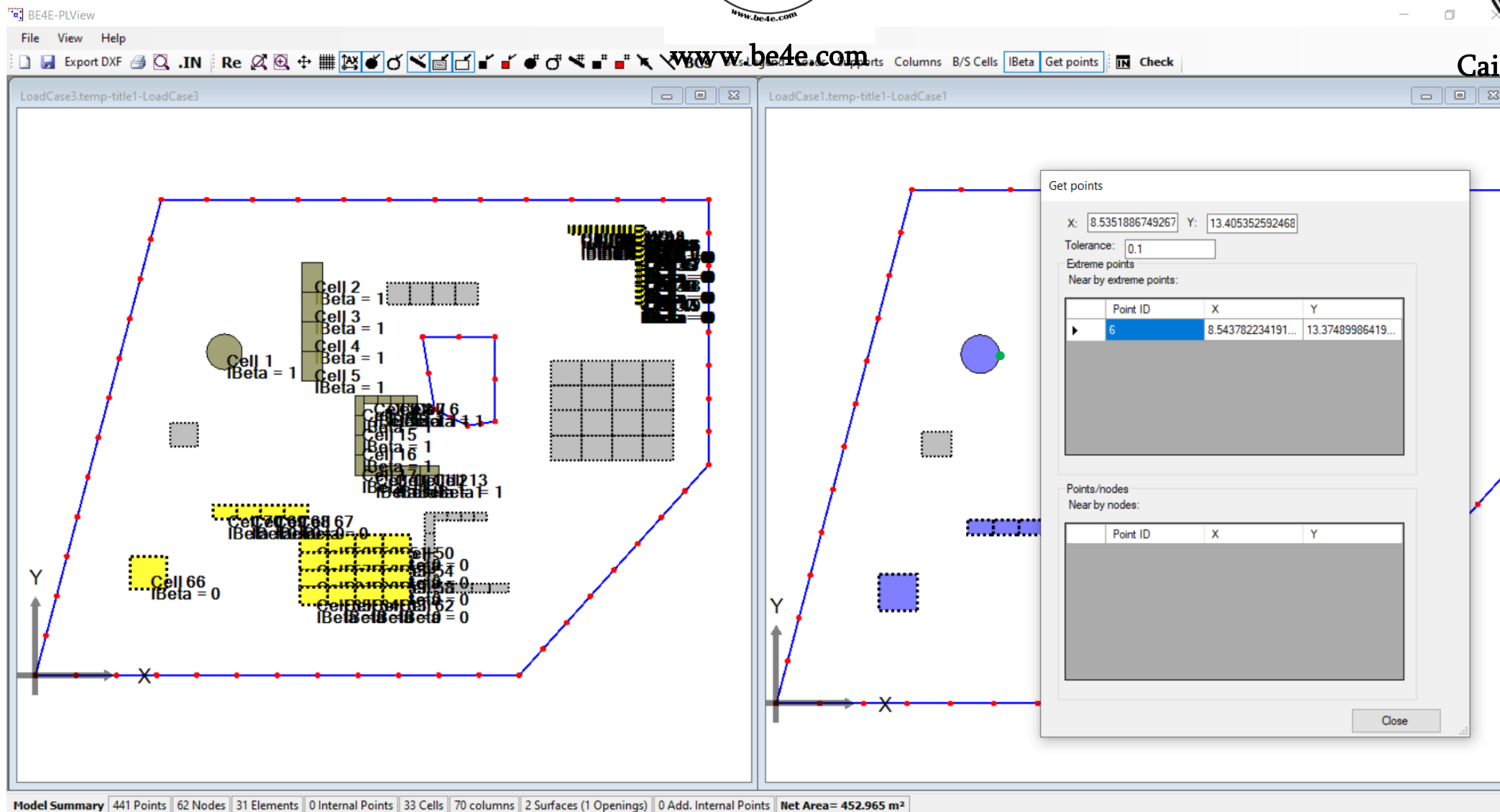
BE model showing BE number, directions, and normal directions

BE model showing boundary conditions



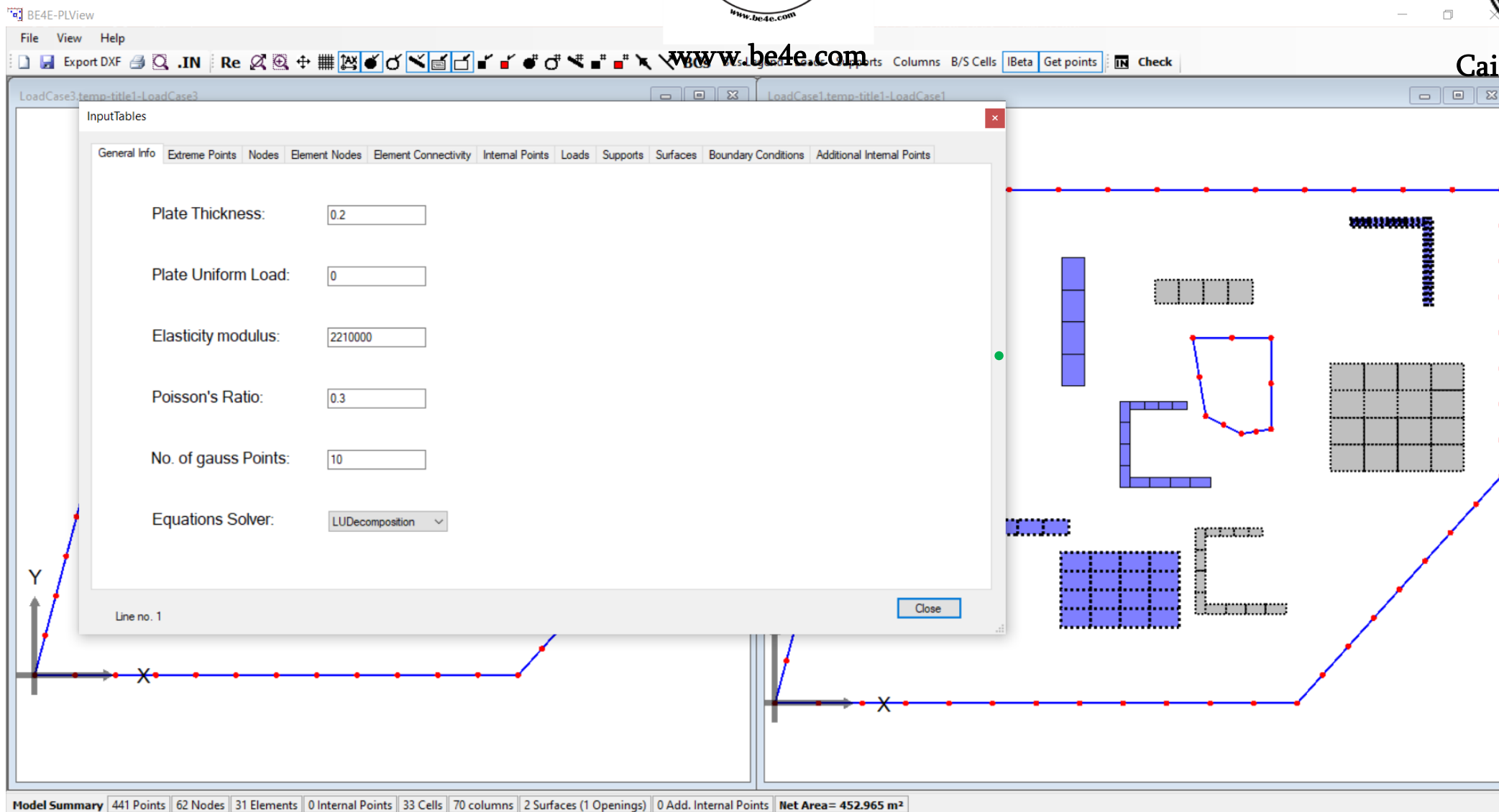
BE model showing load values on loading cells

BE model showing stiffness values on supporting cells



BE model showing IBeta values on supporting cells

BE model showing clicked extreme point coordinates



The screenshot displays the BE4E-PLView software interface. The main window shows a finite element model of a plate with a central hole and several rectangular cutouts. The model is discretized with a mesh of elements, and the boundary is defined by a series of nodes connected by lines. The software interface includes a menu bar (File, View, Help), a toolbar with various icons, and a status bar at the bottom. The status bar displays the following information: Model Summary, 441 Points, 62 Nodes, 31 Elements, 0 Internal Points, 33 Cells, 70 columns, 2 Surfaces (1 Openings), 0 Add. Internal Points, and Net Area = 452.965 m².

The 'InputTables' dialog box is open, showing the following parameters:

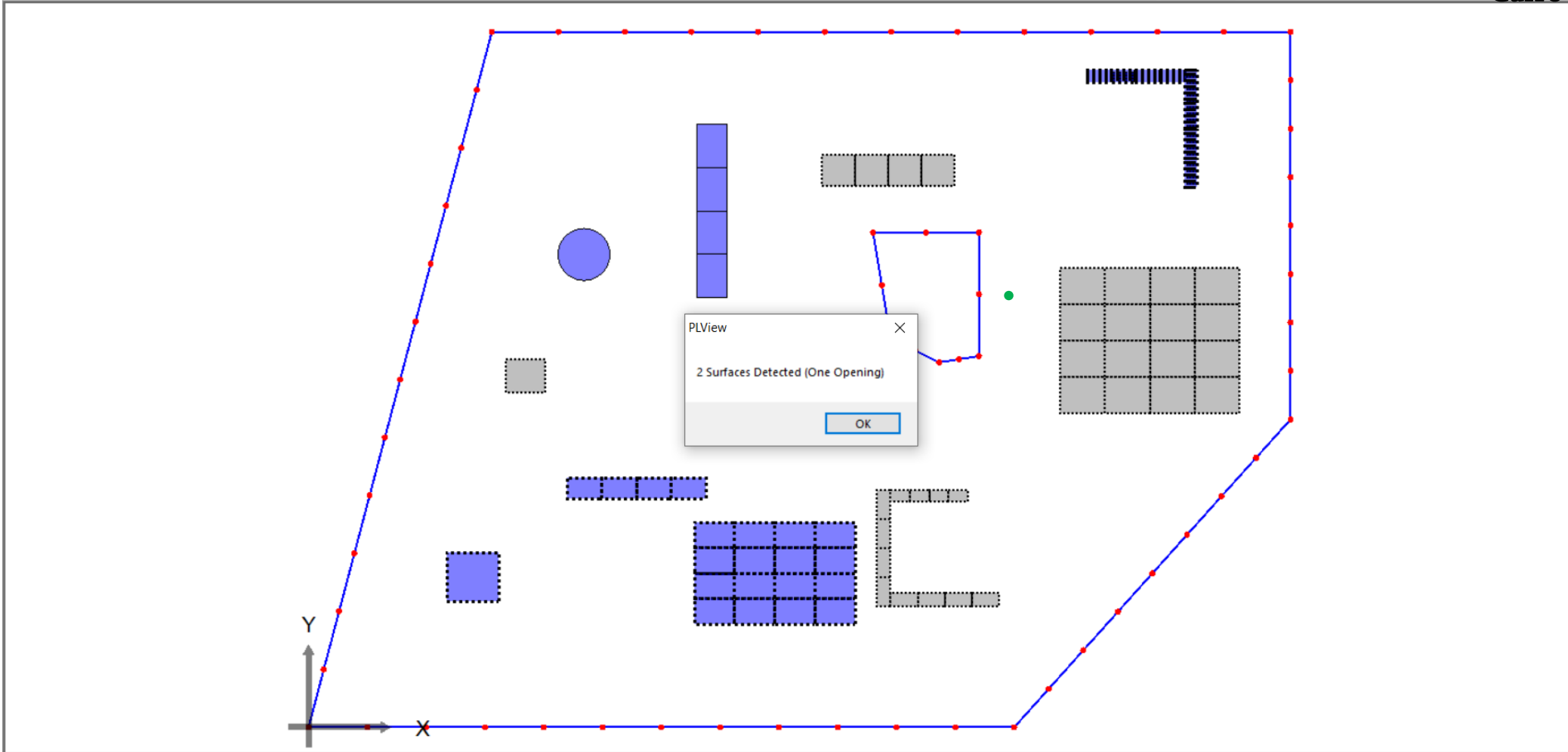
Parameter	Value
Plate Thickness:	0.2
Plate Uniform Load:	0
Elasticity modulus:	2210000
Poisson's Ratio:	0.3
No. of gauss Points:	10
Equations Solver:	LUdecomposition

Tabulated values of the model

BE4E-PLView - [LoadCase1.temp-title1-LoadCase1]

File View Help

Export DXF .IN Re [Tools] www.be4e.com Supports Columns B/S Cells |Beta Get points Check



Model Summary 441 Points 62 Nodes 31 Elements 0 Internal Points 33 Cells 70 columns 2 Surfaces (1 Openings) 0 Add. Internal Points Net Area= 452.965 m²

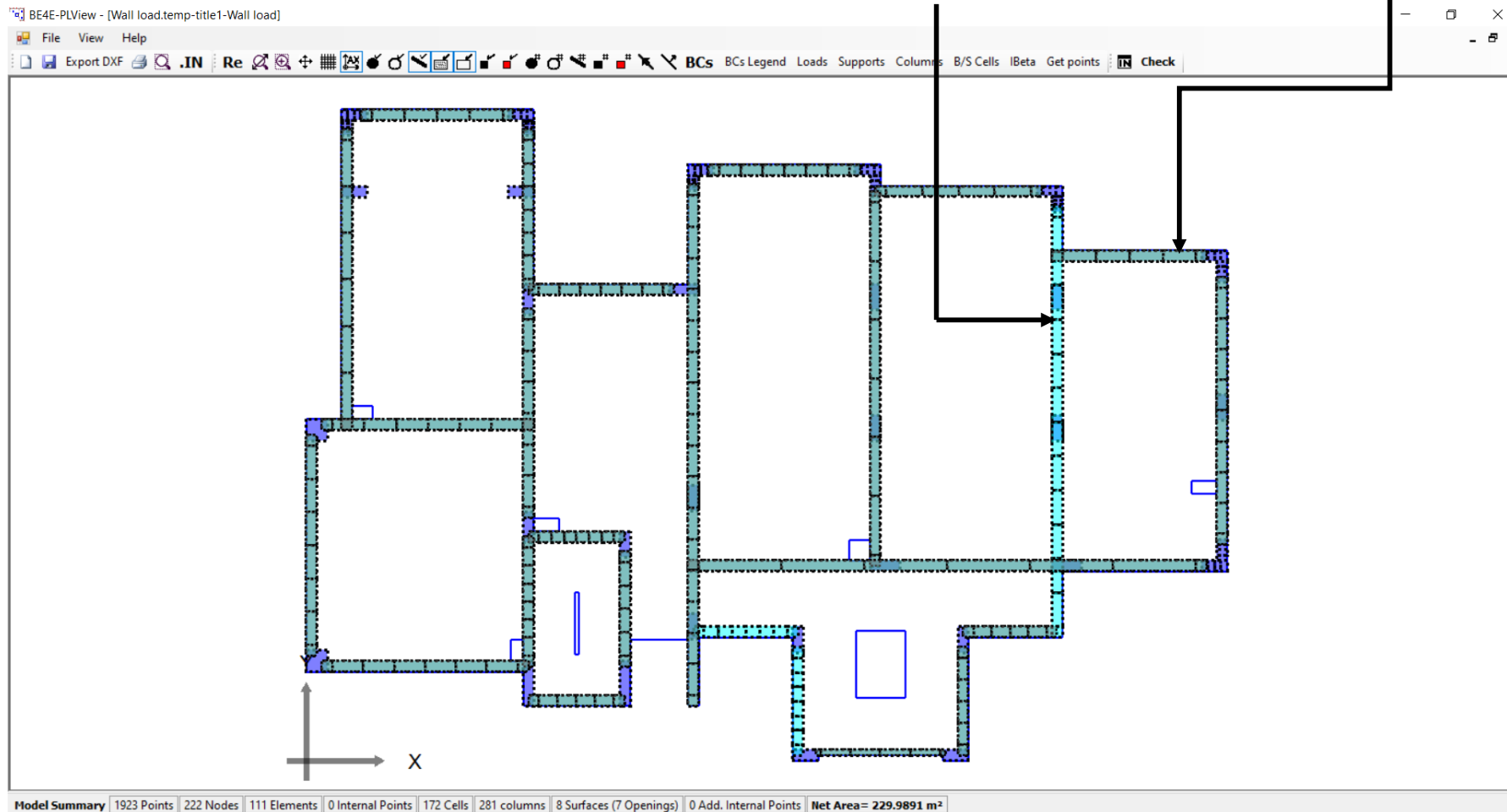
Number of surfaces in the model by clicking the Check button

CUFE-BE



Cairo University

- In the PLView the user can see the number of division for each beam.
- In case of the beam color is light green, it means that it doesn't contain beam own weight.
- In case the beam color is Dark green, it means that it contains beam own weight.



CUFE-BE



www.be4e.com



Cairo University

- A. PLGen – Model generator** ✓
- B. PLView – Numerical model** ✓
- C. PLCoreMan – Manager and solver
- D. PLPost – Post processing
- E. PLDesign – Design tool
- F. PLPAK modelling capabilities

CUFE-BE



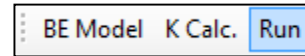
Cairo University

- A. PLGen – Model generator
- B. PLView – Numerical model
- C. PLCoreMan – Manager and solver**
- D. PLPost – Post processing
- E. PLDesign – Design tool
- F. PLPAK modelling capabilities

- PLCoreMan has several tasks:

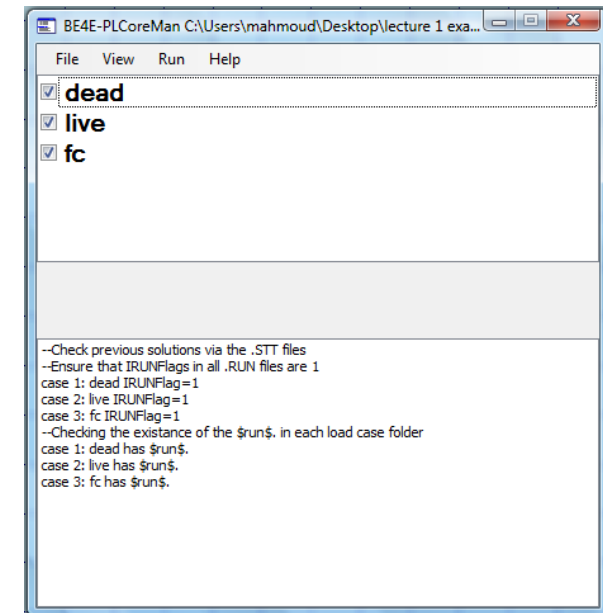
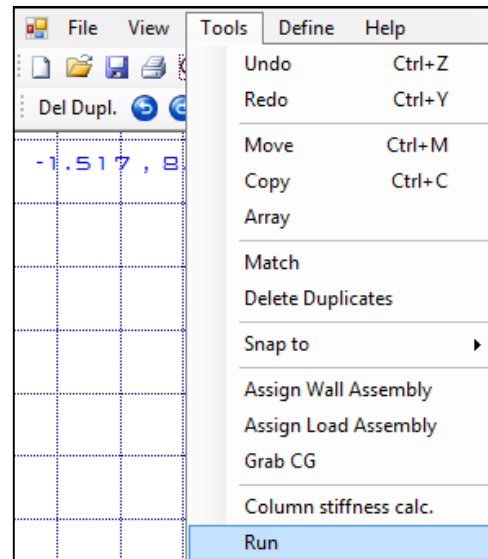
1- Run the model.

2- Transfer between PLPAK components (PLView & PLPost) or between other packages (PLDesign & EHSPAK & P-PPAK & PTPAK & NLPAK).



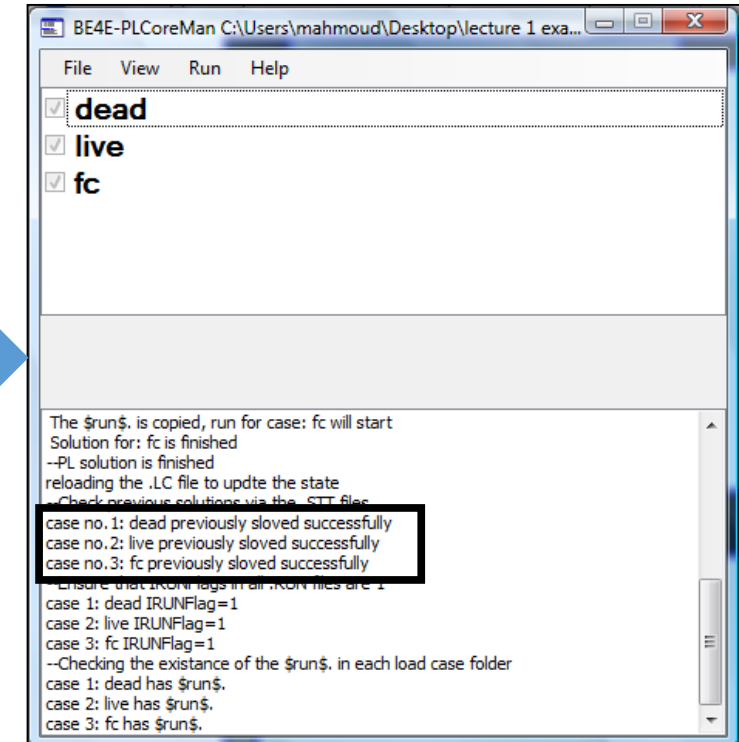
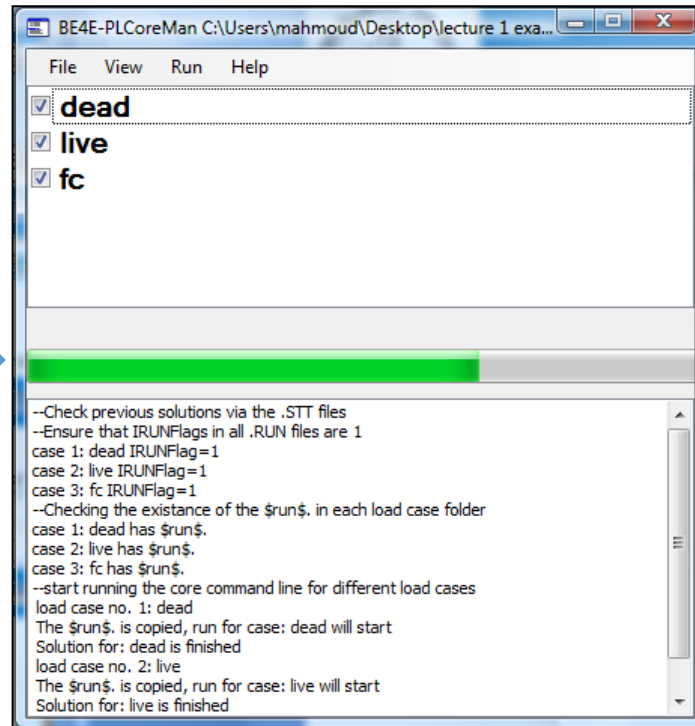
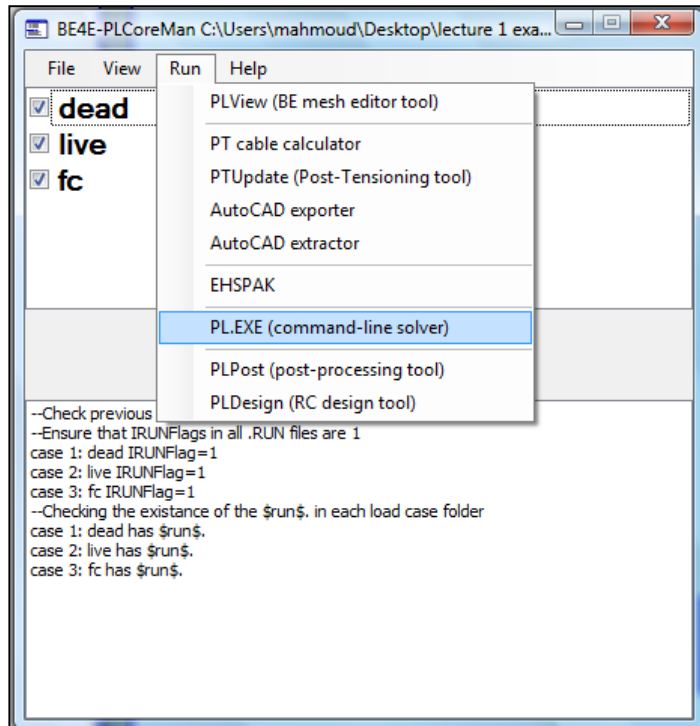
3- Open PL controls, manual, and about.

OR

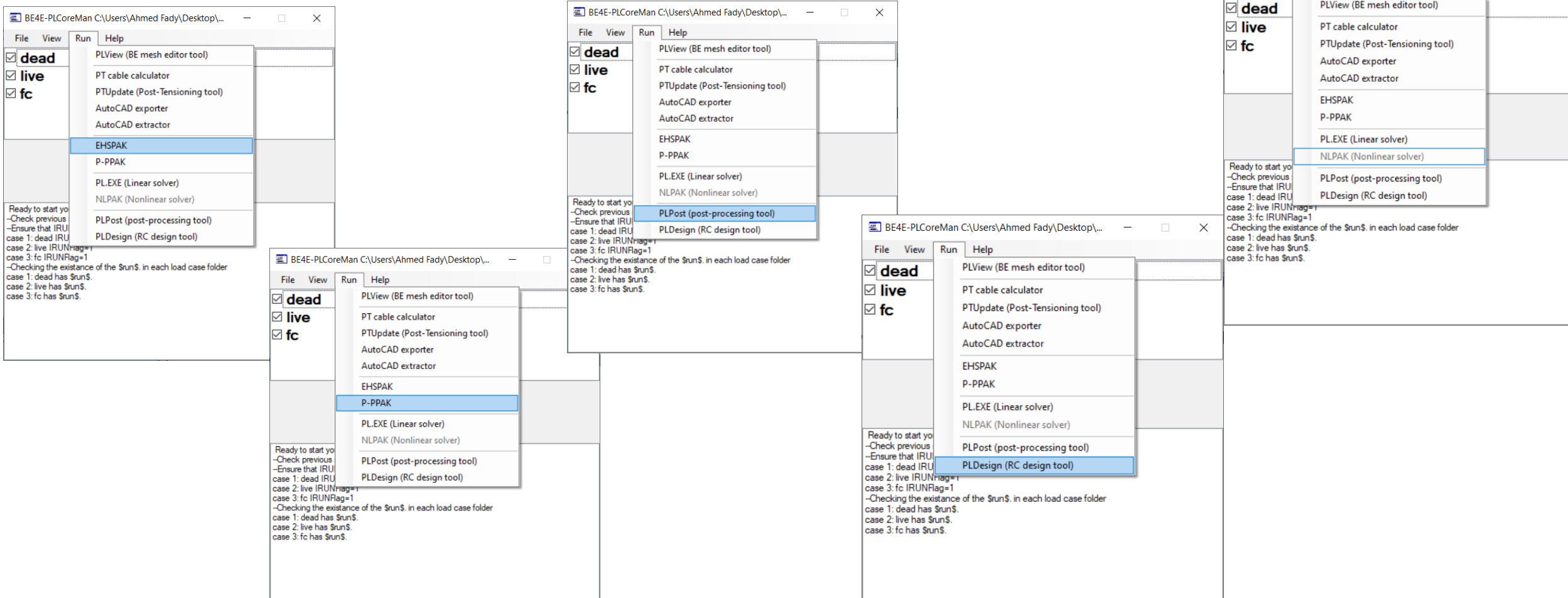


1- Run the model

- The user should see in the text window that the load cases run successfully.
- If the load cases didn't run successfully, model must be checked again in PLView.



2- Transfer between PLPAK components (PLView & PLPost) or between other packages (PLDesign & EHSPAK & P-PPAK & PTPAK & NLPK)



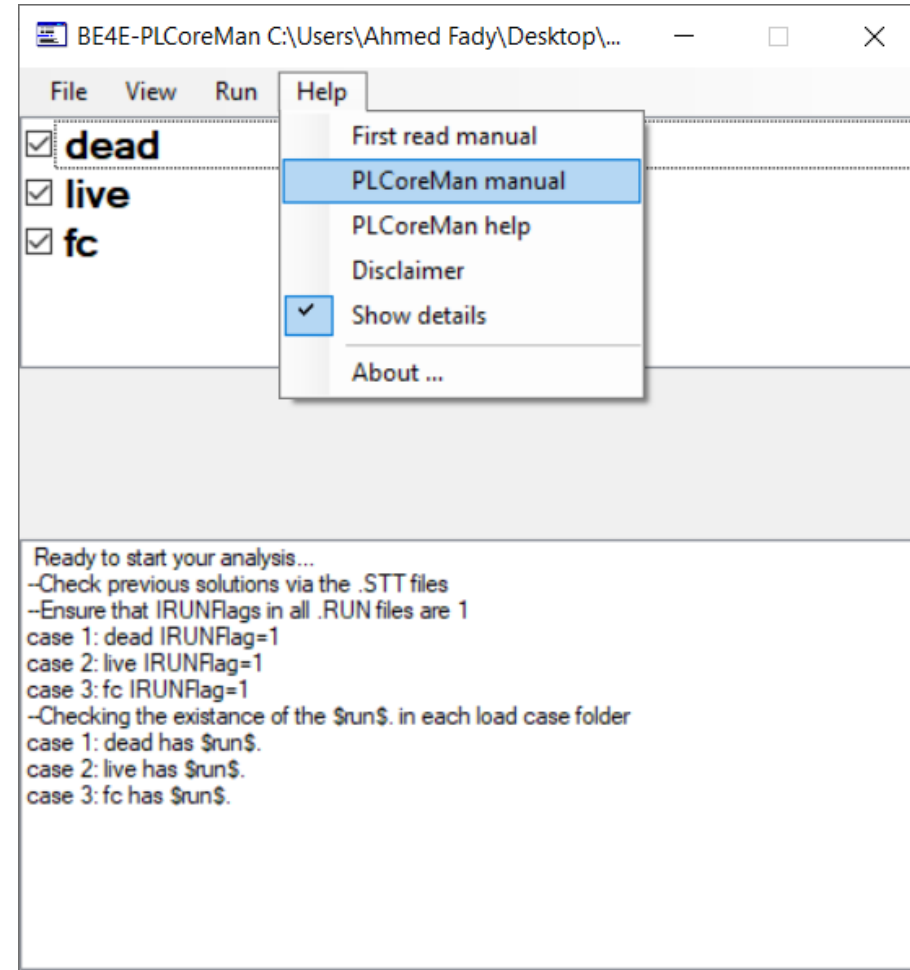
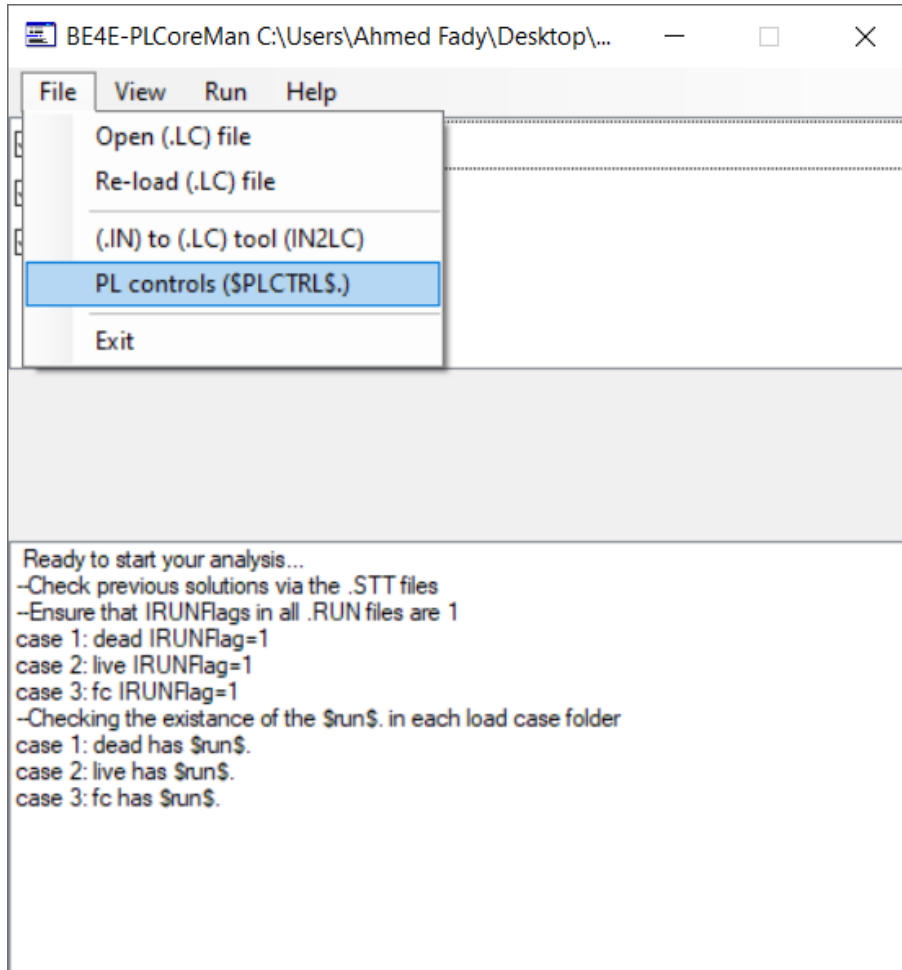
The image displays four screenshots of the BE4E-PLCoreMan software interface, illustrating the process of transferring between different PLPAK components. Each screenshot shows the 'Run' menu with various tool options, and the 'live' and 'fc' components are checked in the left sidebar.

- Top Left Screenshot:** The 'Run' menu is open, and 'EHSPAK' is highlighted. The status bar at the bottom indicates: "Ready to start yo... -Check previous -Ensure that IRU... case 1: dead IRU... case 2: live IRUNflag=1 case 3: fc IRUNflag=1 -Checking the existence of the \$run\$. in each load case folder case 1: dead has \$run\$. case 2: live has \$run\$. case 3: fc has \$run\$."
- Top Middle Screenshot:** The 'Run' menu is open, and 'PLPost (post-processing tool)' is highlighted. The status bar at the bottom indicates: "Ready to start yo... -Check previous -Ensure that IRU... case 1: dead IRU... case 2: live IRUNflag=1 case 3: fc IRUNflag=1 -Checking the existence of the \$run\$. in each load case folder case 1: dead has \$run\$. case 2: live has \$run\$. case 3: fc has \$run\$."
- Bottom Left Screenshot:** The 'Run' menu is open, and 'P-PPAK' is highlighted. The status bar at the bottom indicates: "Ready to start yo... -Check previous -Ensure that IRU... case 1: dead IRU... case 2: live IRUNflag=1 case 3: fc IRUNflag=1 -Checking the existence of the \$run\$. in each load case folder case 1: dead has \$run\$. case 2: live has \$run\$. case 3: fc has \$run\$."
- Bottom Right Screenshot:** The 'Run' menu is open, and 'PLDesign (RC design tool)' is highlighted. The status bar at the bottom indicates: "Ready to start yo... -Check previous -Ensure that IRU... case 1: dead IRU... case 2: live IRUNflag=1 case 3: fc IRUNflag=1 -Checking the existence of the \$run\$. in each load case folder case 1: dead has \$run\$. case 2: live has \$run\$. case 3: fc has \$run\$."

On the right side of the image, a larger screenshot shows the 'Run' menu with 'NLPK (Nonlinear solver)' highlighted. The status bar at the bottom indicates: "Ready to start yo... -Check previous -Ensure that IRU... case 1: dead IRU... case 2: live IRUNflag=1 case 3: fc IRUNflag=1 -Checking the existence of the \$run\$. in each load case folder case 1: dead has \$run\$. case 2: live has \$run\$. case 3: fc has \$run\$."

3- Open PL controls, manual, and about

www.be4e.com



CUFE-BE



www.be4e.com



Cairo University

- A. PLGen – Model generator** ✓
- B. PLView – Numerical model** ✓
- C. PLCoreMan – Manager and solver** ✓
- D. PLPost – Post processing**
- E. PLDesign – Design tool**
- F. PLPAK modelling capabilities**

CUFE-BE



www.be4e.com



Cairo University

- A. PLGen – Model generator
- B. PLView – Numerical model
- C. PLCoreMan – Manager and solver
- D. PLPost – Post processing**
- E. PLDesign – Design tool
- F. PLPAK modelling capabilities

The PLPost is the post-processing tool to demonstrate analysis results. PLPost can be categorized into four items:

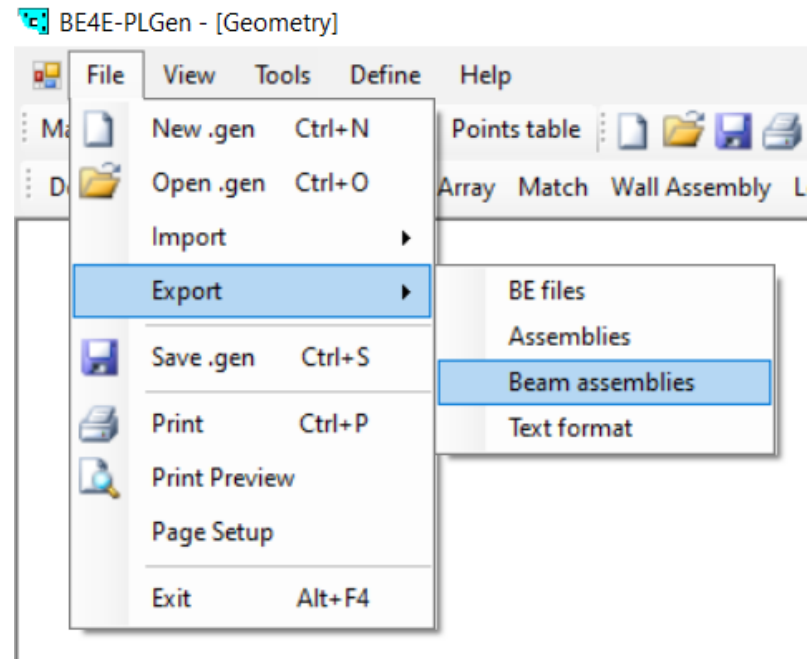
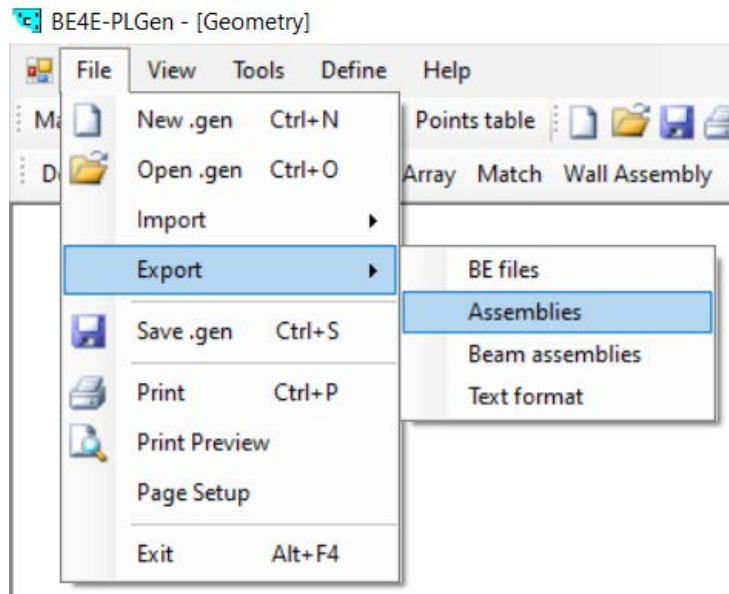
1. **Files exported from PLGen**
2. **Load combinations**
3. **Slab results**
 - 3.1. **Contour results**
 - 3.2. **Query result**
 - 3.3. **Strip results**
 - 3.4. **Supporting elements results**
4. **Beam results**

1. Files exported from PLGen

There are cases that user have to export file from PLGen before using PLPost:

- Export assemblies: this case is used to show the total loads of columns, shear walls and wall assemblies (cores).
- Export beam assemblies: this case is used to show the results of the beams.

Restore these files in the PLPost will be demonstrated later.

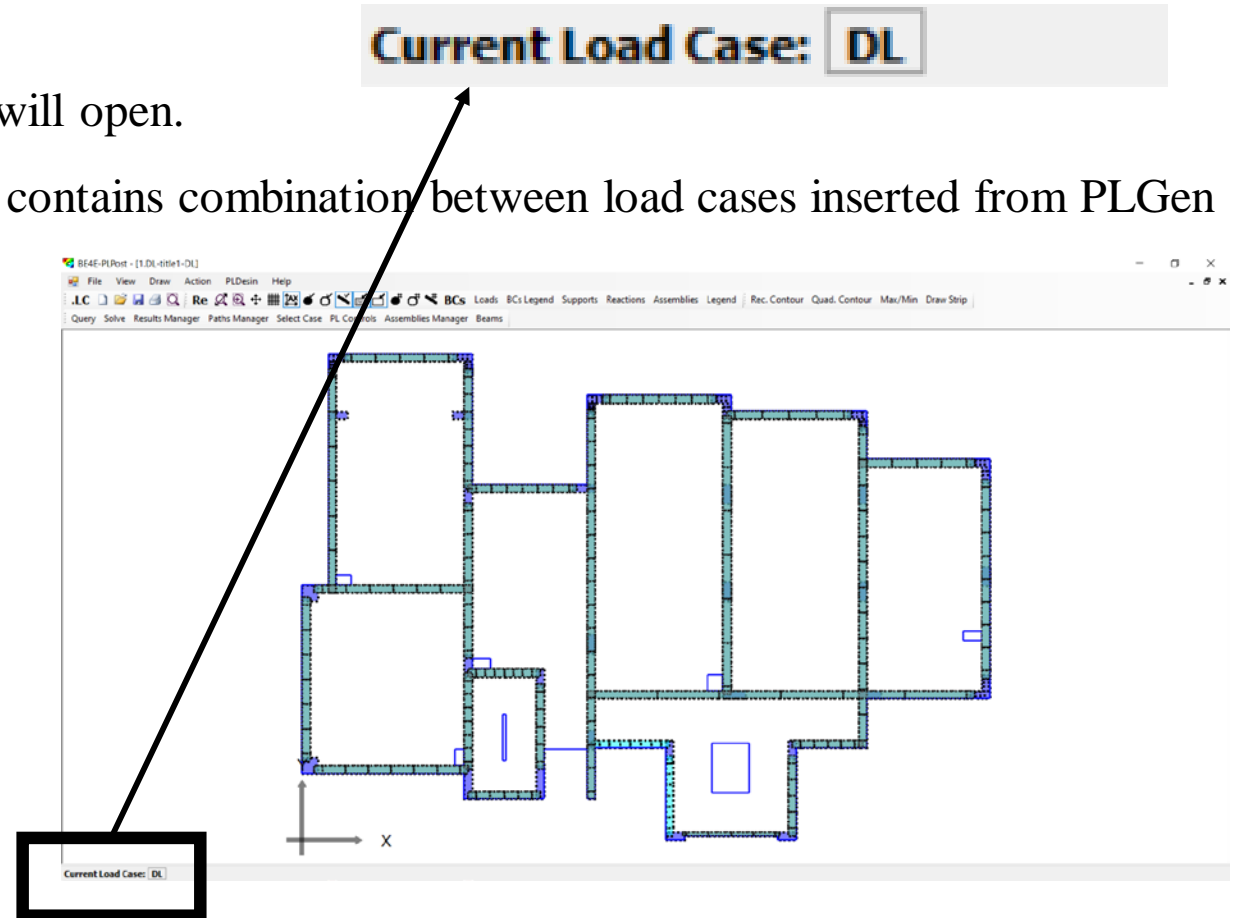
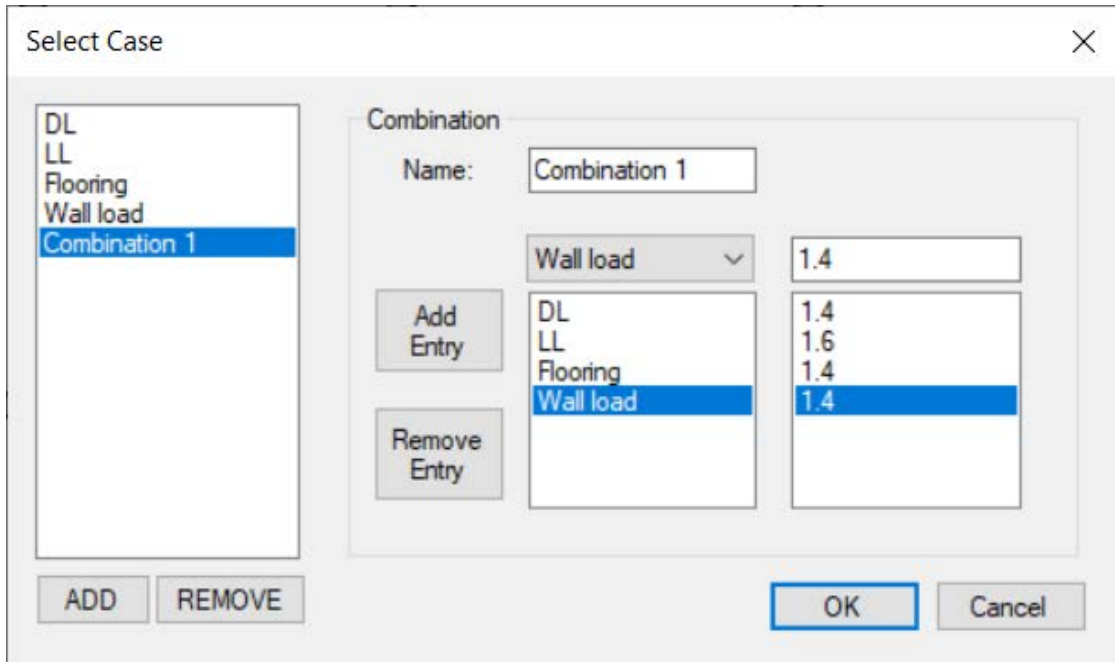


2. Load combinations

The lower tabs of the PLPost contain by default current load case (by default it is the first load case), can be changed by double click on it.

If the user press double click on it load combinations window will open.

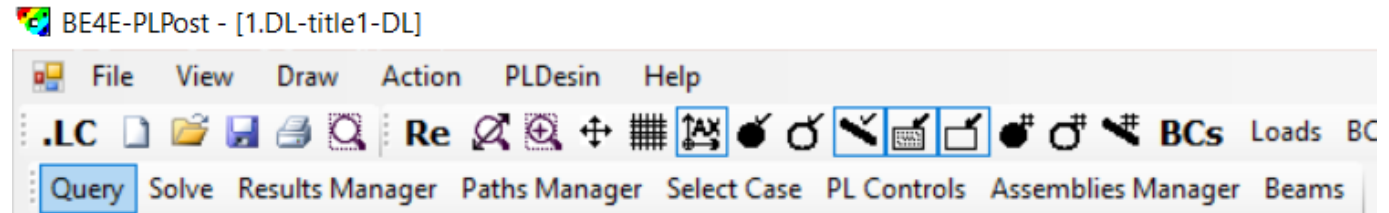
The user can add cases like ultimate, working cases, each case contains combination between load cases inserted from PLGen



3. Slab results

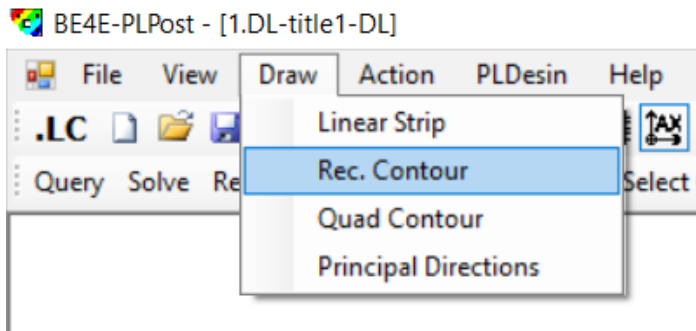
There are 3 types for showing results in slab:

- Contour (main, quadratic, rectangular)
- Strip
- Query for specific point



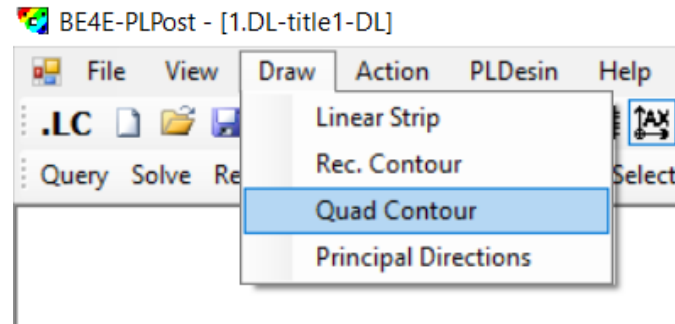
Draw rectangular contour

Rec. Contour Quad. Contour Max/Min Draw Strip



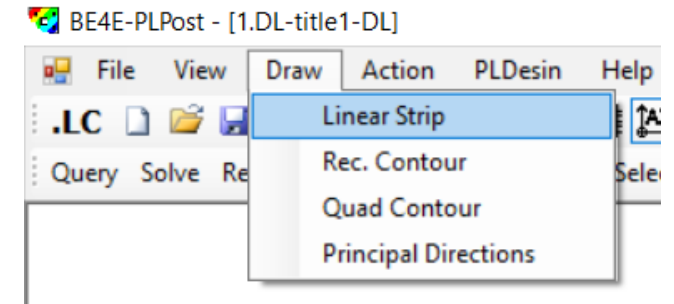
Draw quadratic contour

Rec. Contour Quad. Contour Max/Min Draw Strip



Draw strip

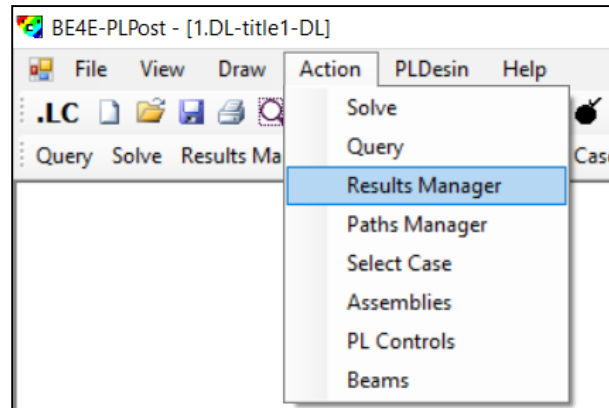
Rec. Contour Quad. Contour Max/Min Draw Strip



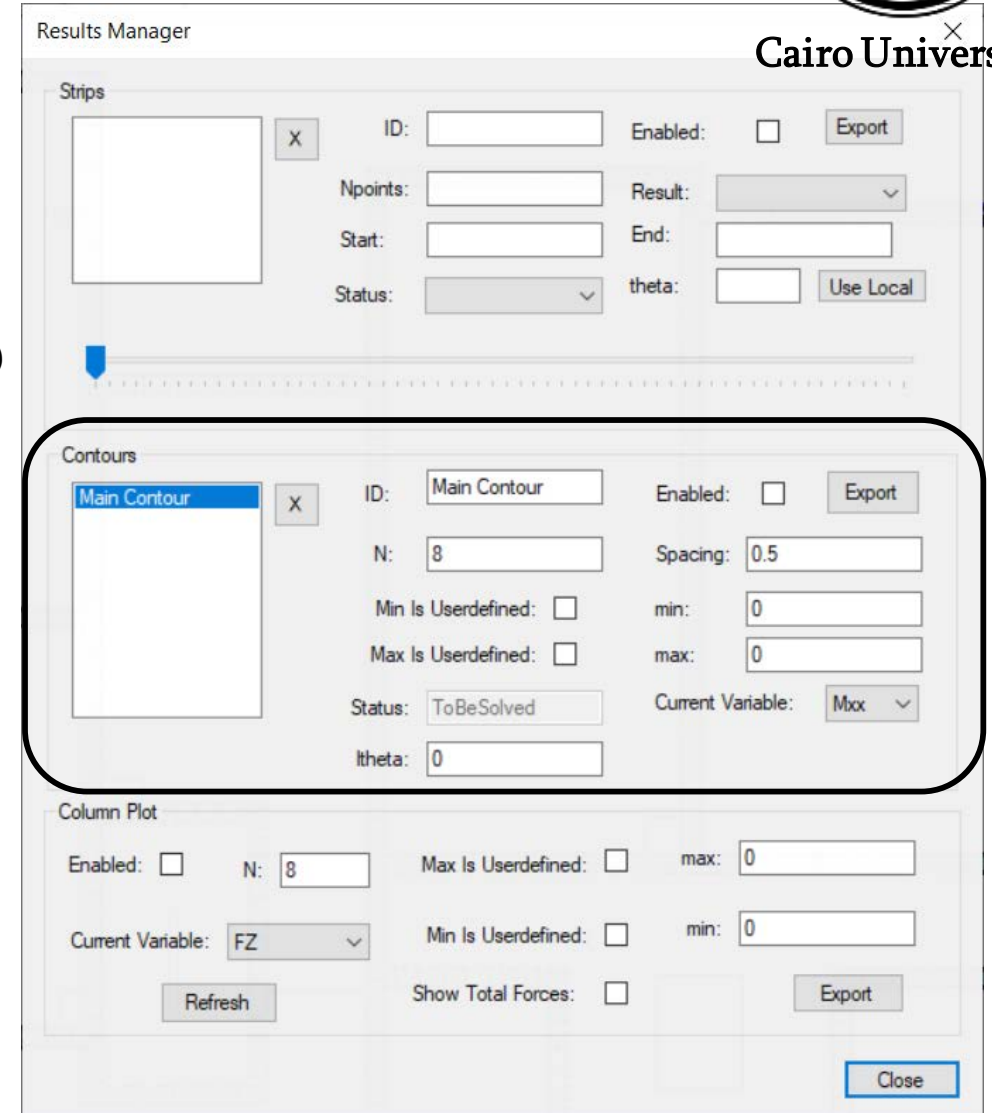
3. Slab results – contour results

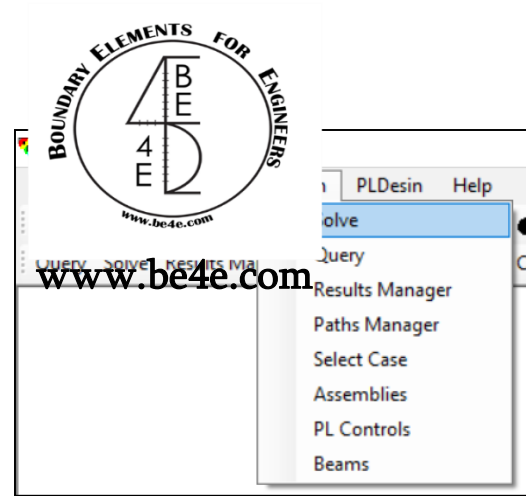
Straining action for main contour using result manager:

- Open the results manager
- The results manager consists of three parts at the middle part (Contours) we have the Main Contour.
- Mark the check box (Enabled).
- Insert the spacing between grids.
- Insert number of colors (N).
- Choose the Current Variable
- Insert if there is min/max value for user define.
- Press on Close.



OR



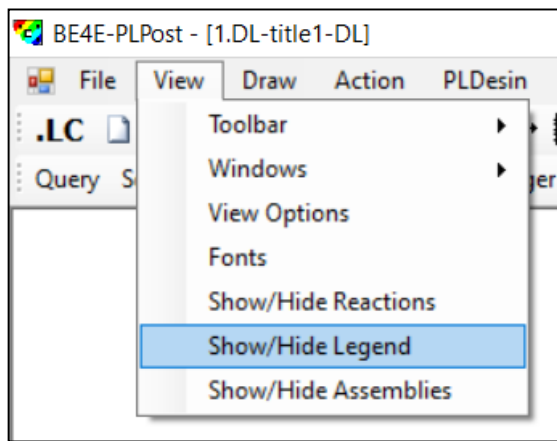
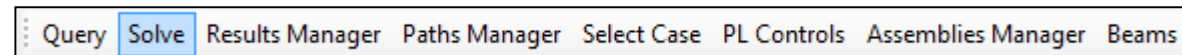


3. Slab results – contour results

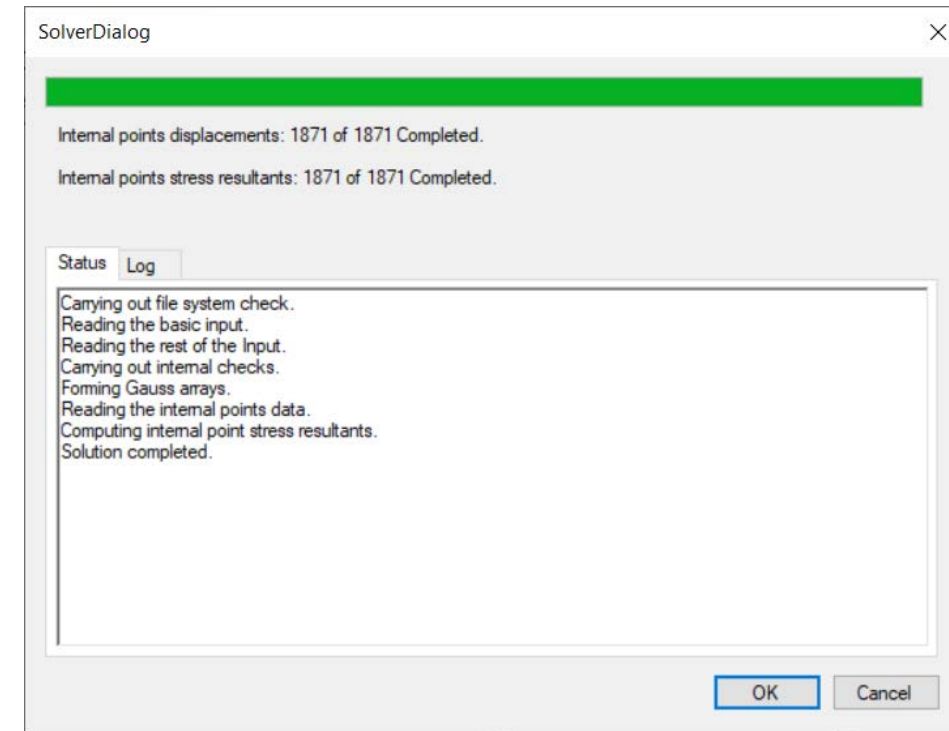
Straining action for main contour using result manager:

- Solve the main contour.
- Solver dialog will appear and after solving press on (F3) or refresh.
- The user can show/hide the legend.

OR



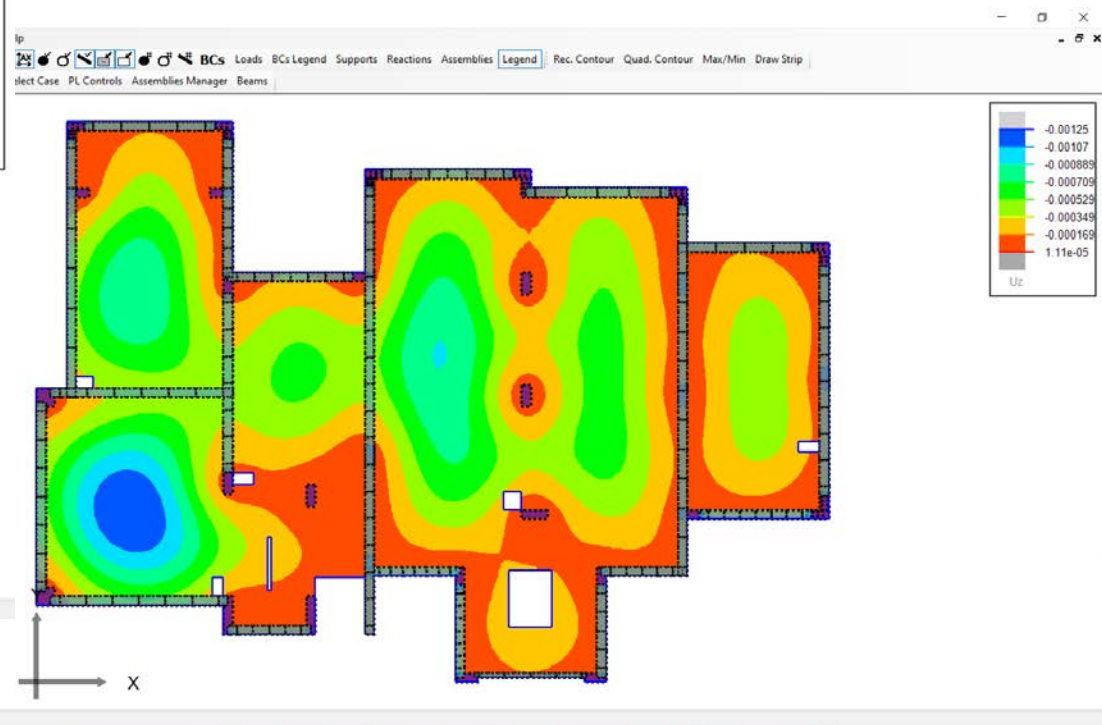
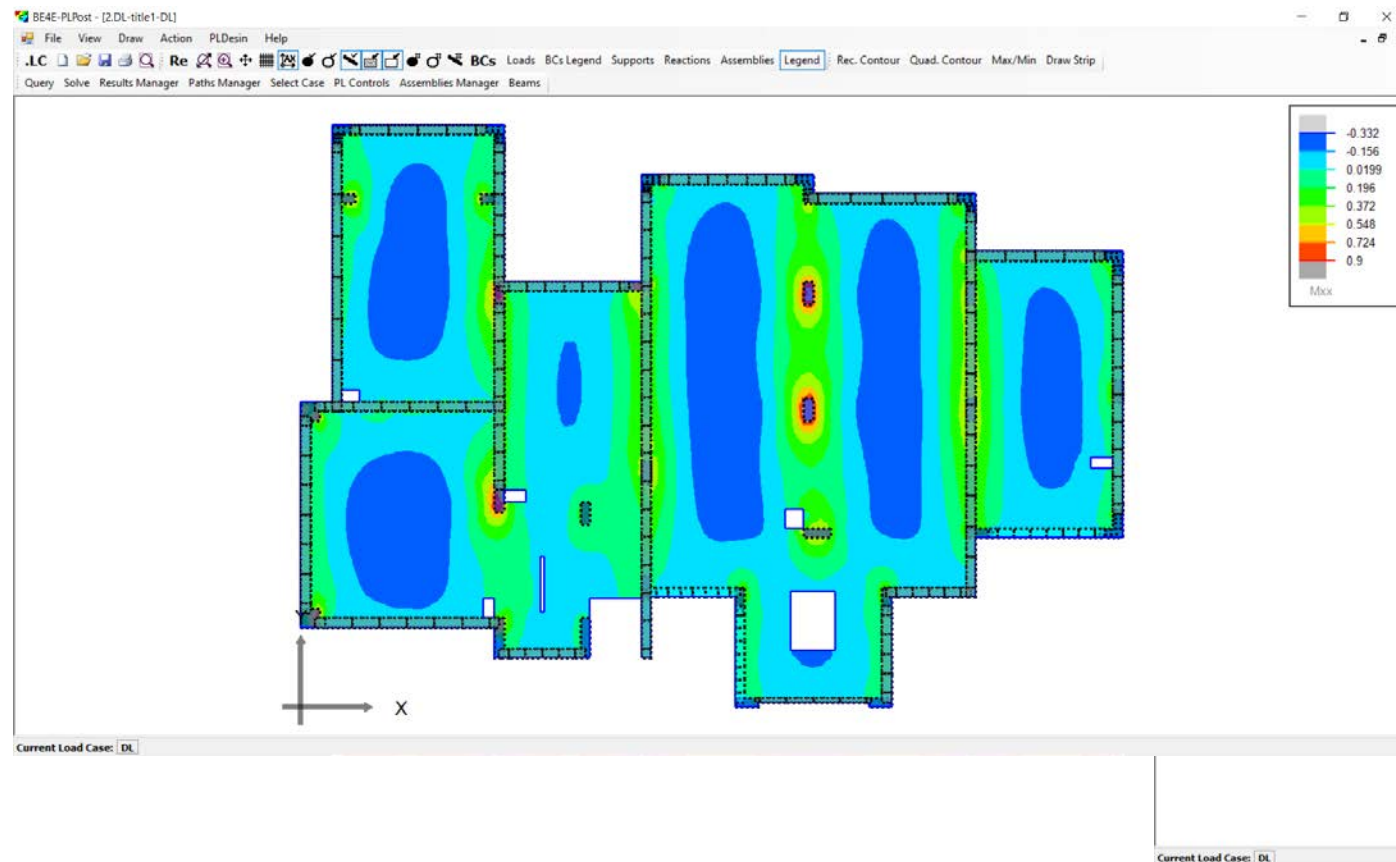
OR



3. Slab results – contour results

Straining action for main contour using result manager:

- The user can see the slab straining actions in any direction, only by changing the current variable on Result Manager.

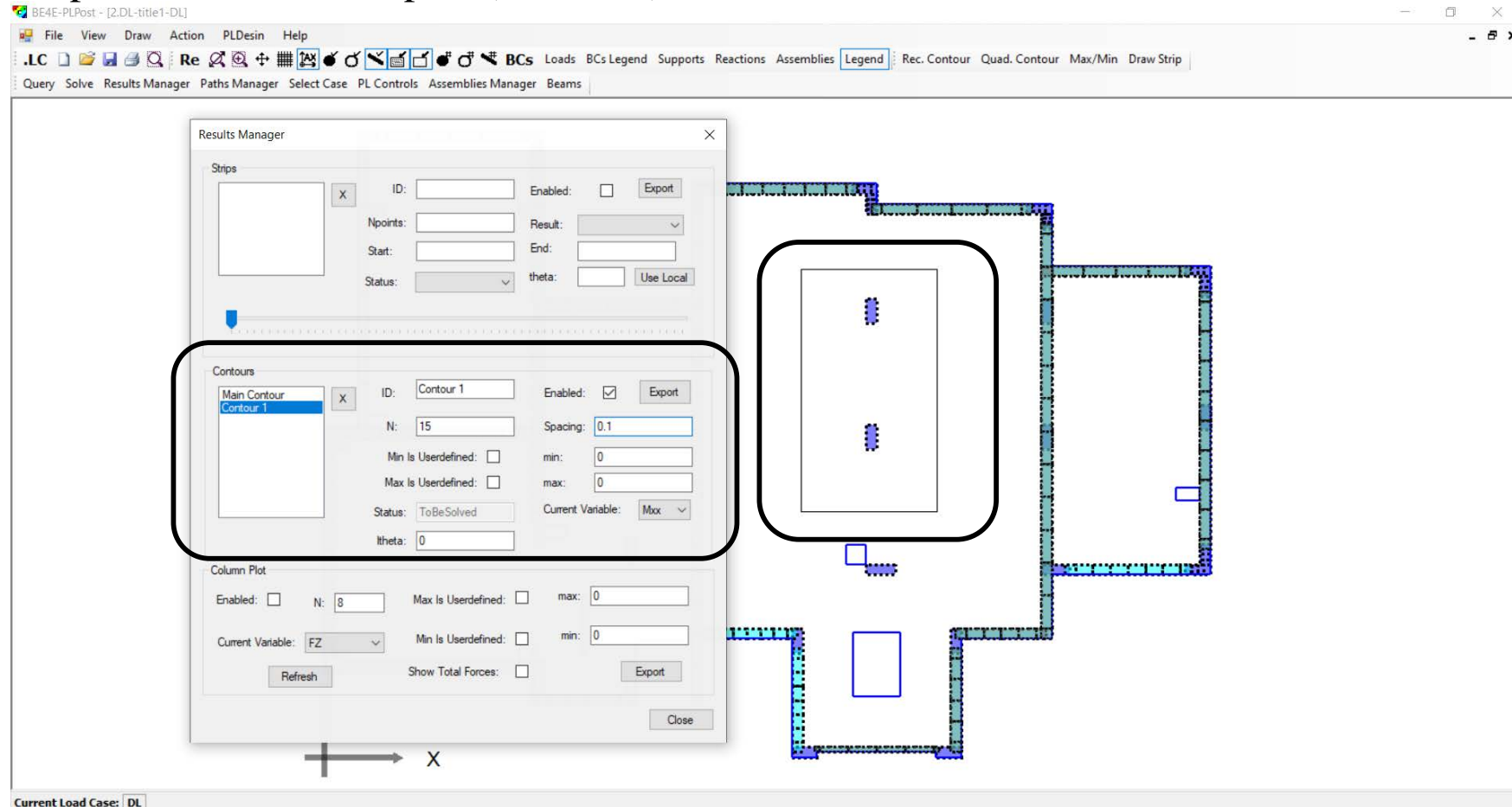


3. Slab results – contour results

www.be4e.com

Straining action for rectangular contour using result manager:

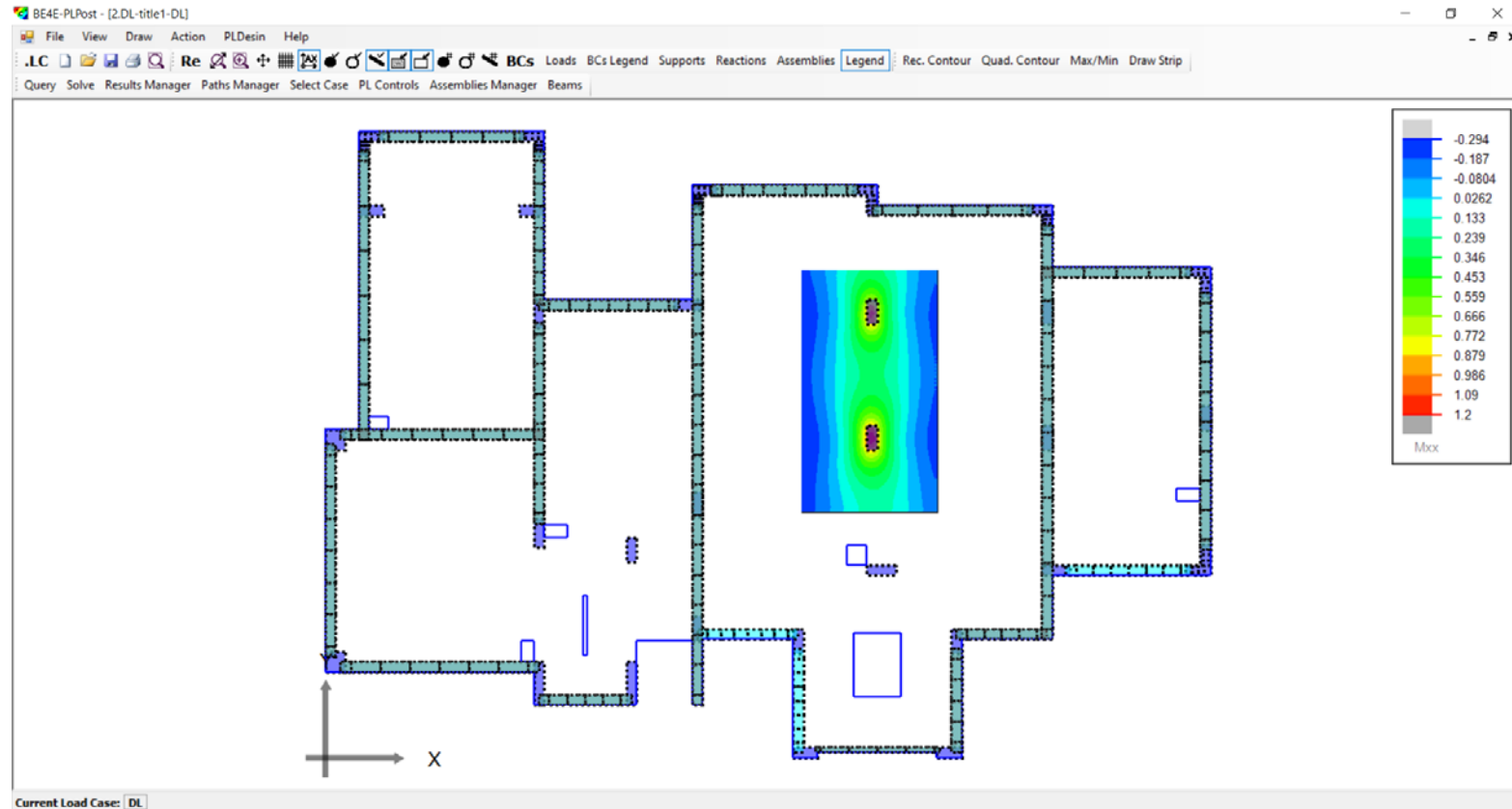
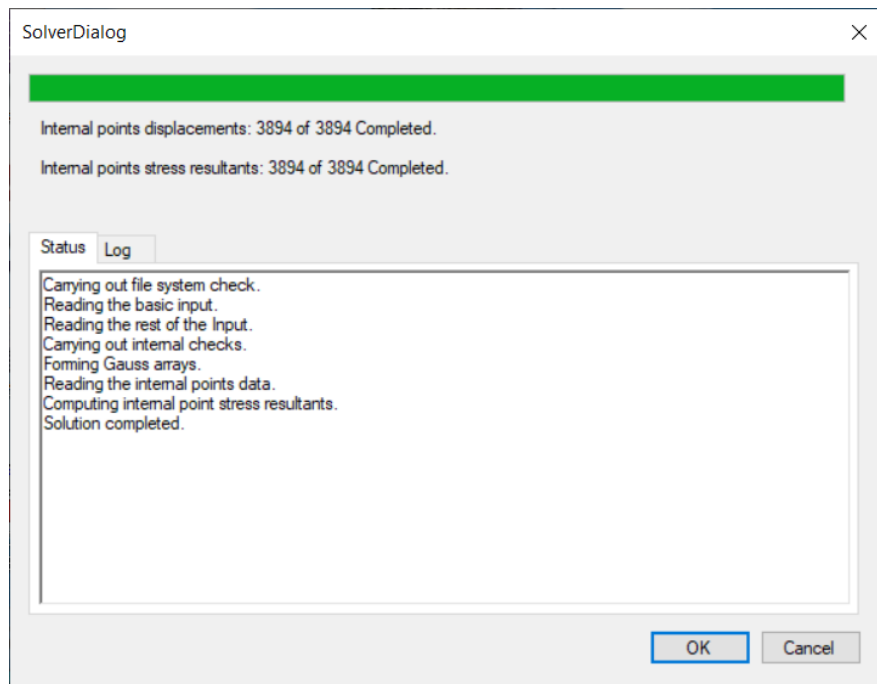
- Draw the rectangular contour.
- The results manager consists of three parts at the middle part (Contours), it has the Main Contour in addition to Contour 1.
- Contour 1 is created and it's automatically enabled.
- Insert the spacing between grids.
- Insert number of colors (N).
- Choose the Current Variable
- Edit the min/max value for user define (if exist).
- Press on Close.
- Solve the contour.



3. Slab results – contour results

Straining action for rectangular contour using result manager:

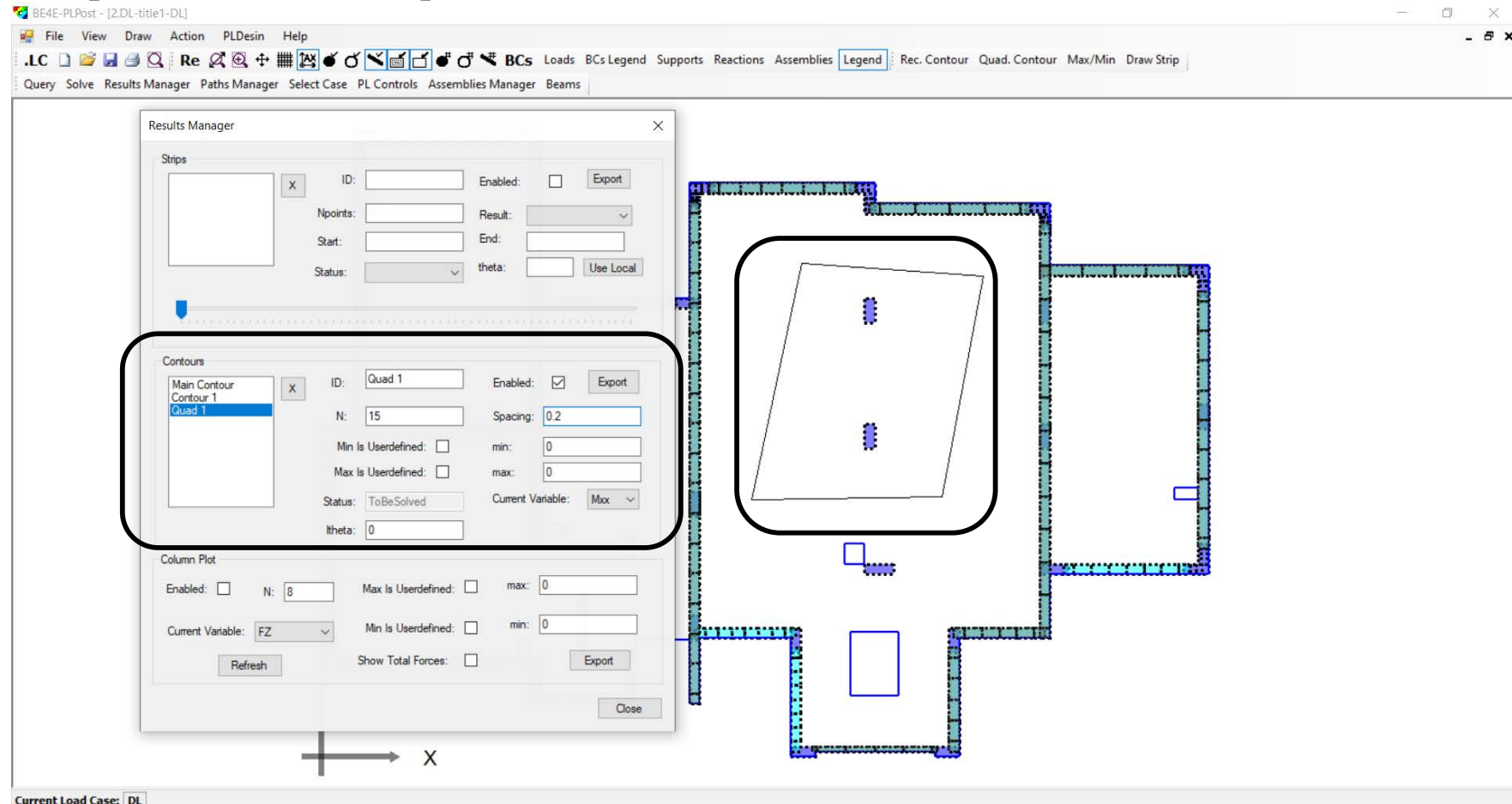
- Solve the main contour.
- Solver dialog will appear and after solving press on (F3) or refresh.
- The user can show/hide the legend.



3. Slab results – contour results

Straining action for quadratic contour using result manager:

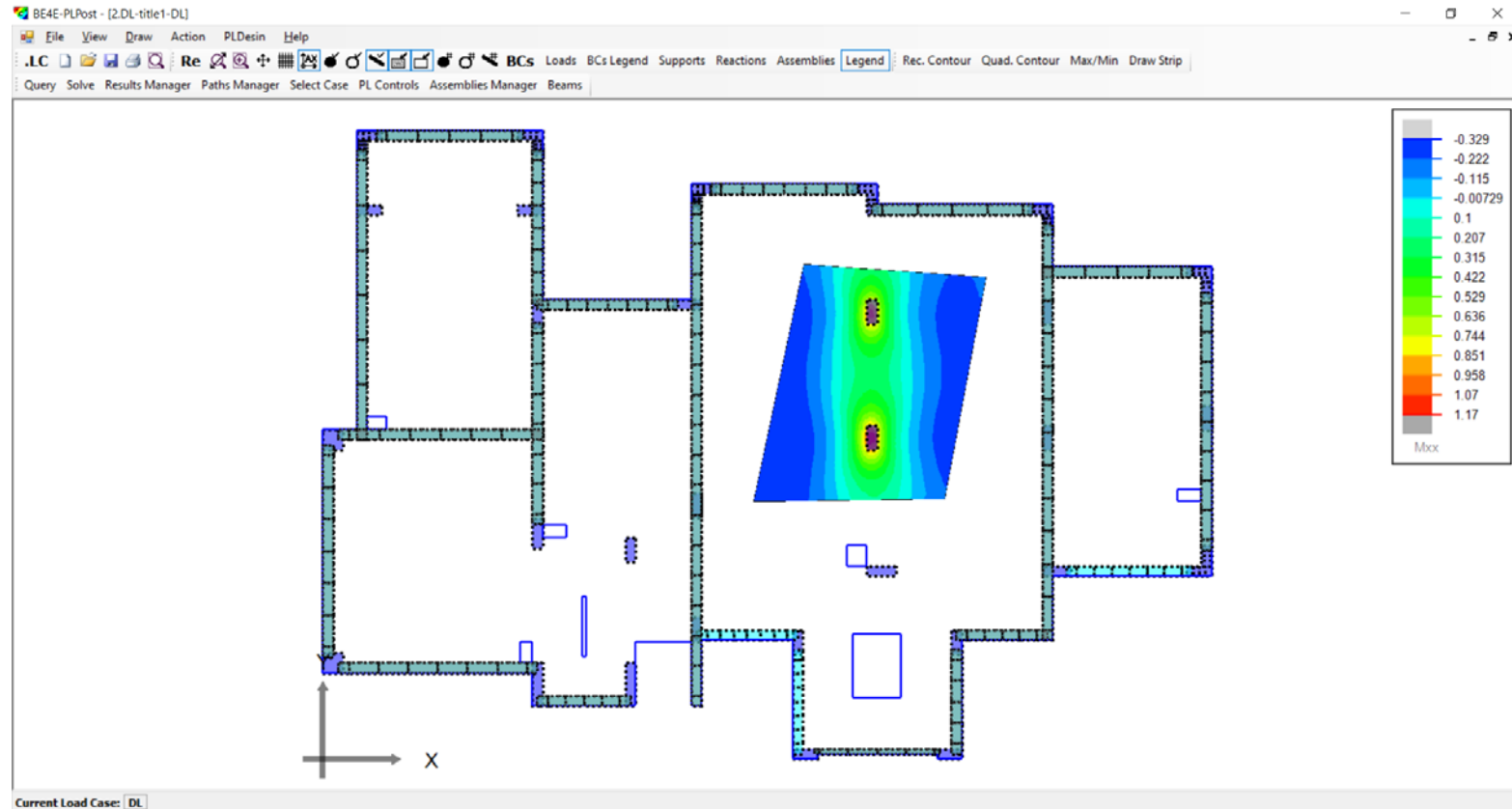
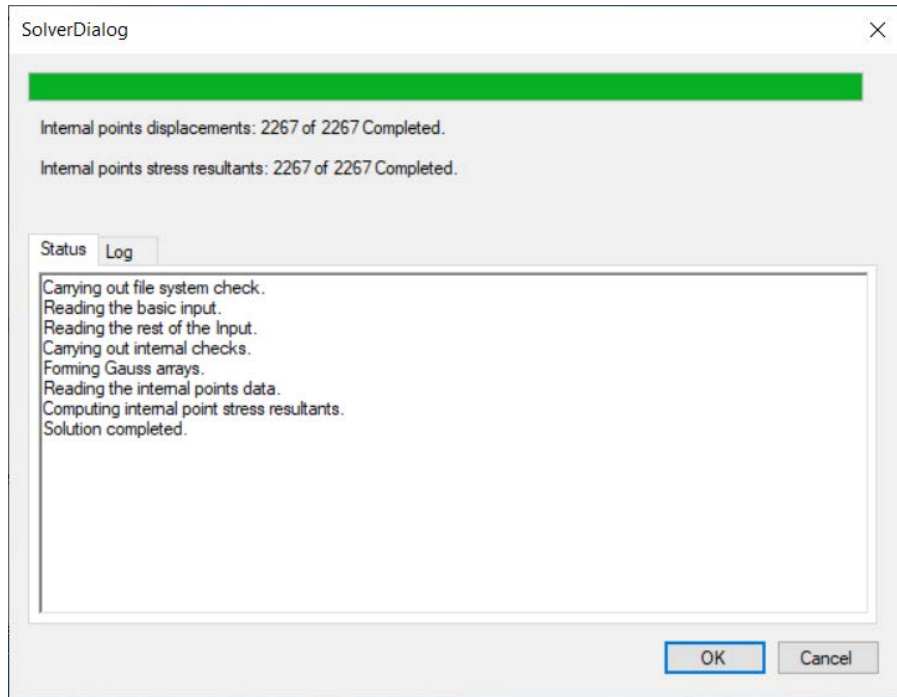
- Draw the quadratic contour.
- The results manager consists of three parts at the middle part (Contours), it has the Main Contour in addition to Contour 1, and quad 1.
- Quad 1 is created and it's automatically enabled.
- Insert the spacing between grids.
- Insert number of colors (N).
- Choose the Current Variable
- Edit the min/max value for user define (if exist).
- Press on Close.
- Solve the contour.



3. Slab results – contour results

Straining action for quadratic contour using result manager:

- Solve the main contour.
- Solver dialog will appear and after solving press on (F3) or refresh.
- The user can show/hide the legend.

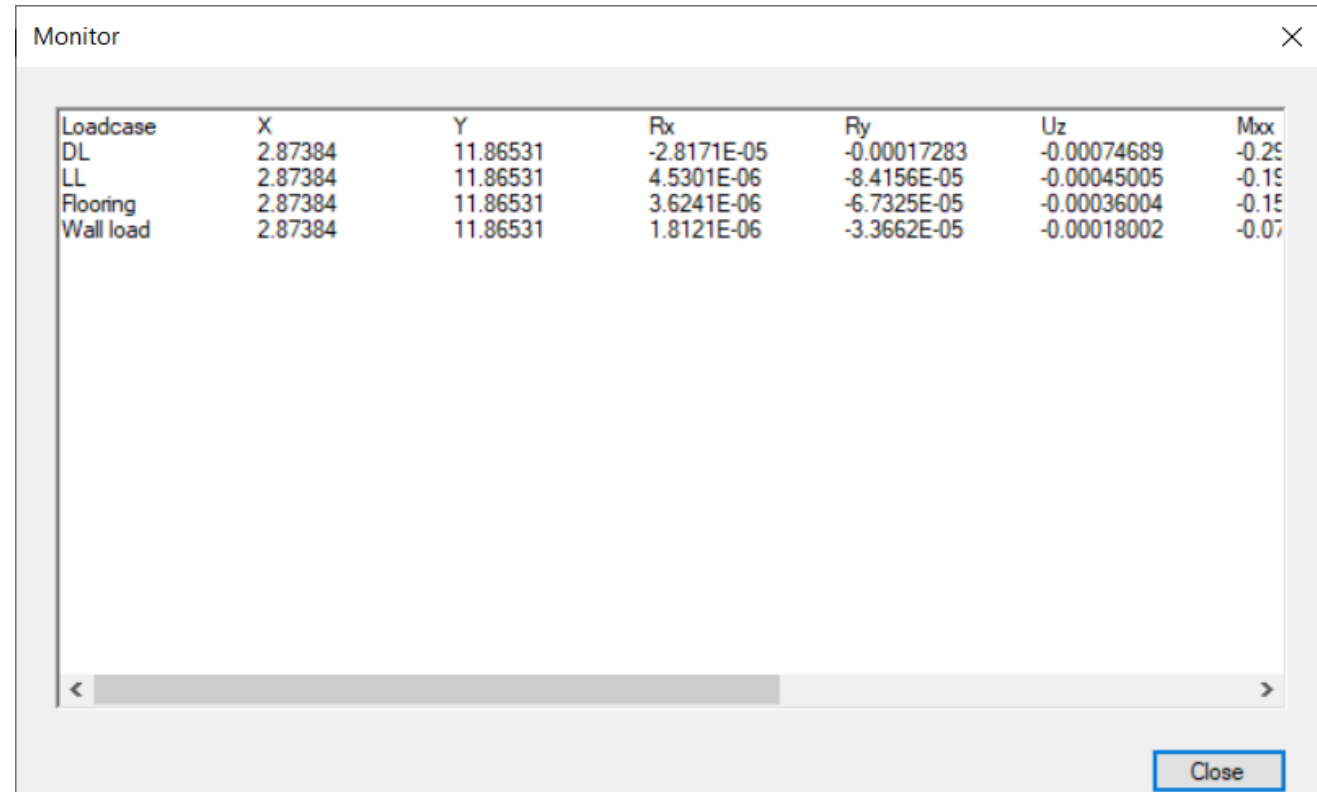
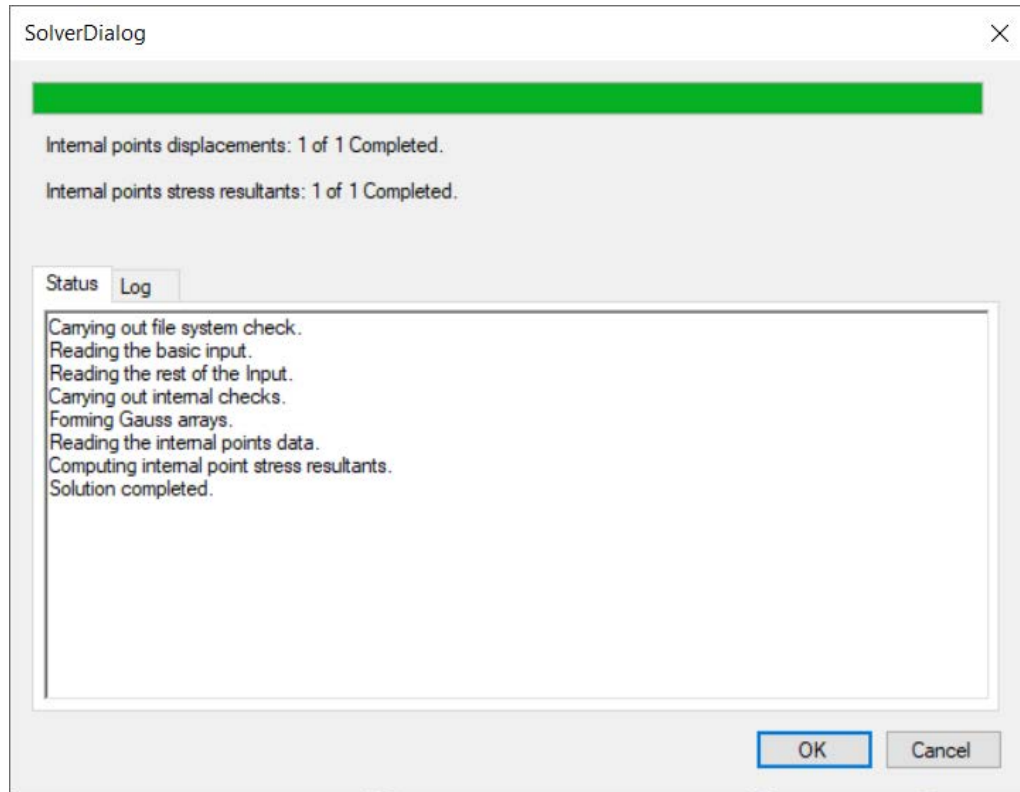


3. Slab results – query result

www.be4e.com

The user can see the straining action at any point by using Query tab:

- After pressing on Query, choose any point to view its straining actions in all cases & their combinations.
- The user doesn't need to go to Result Manager as any previous Contour.



Monitor

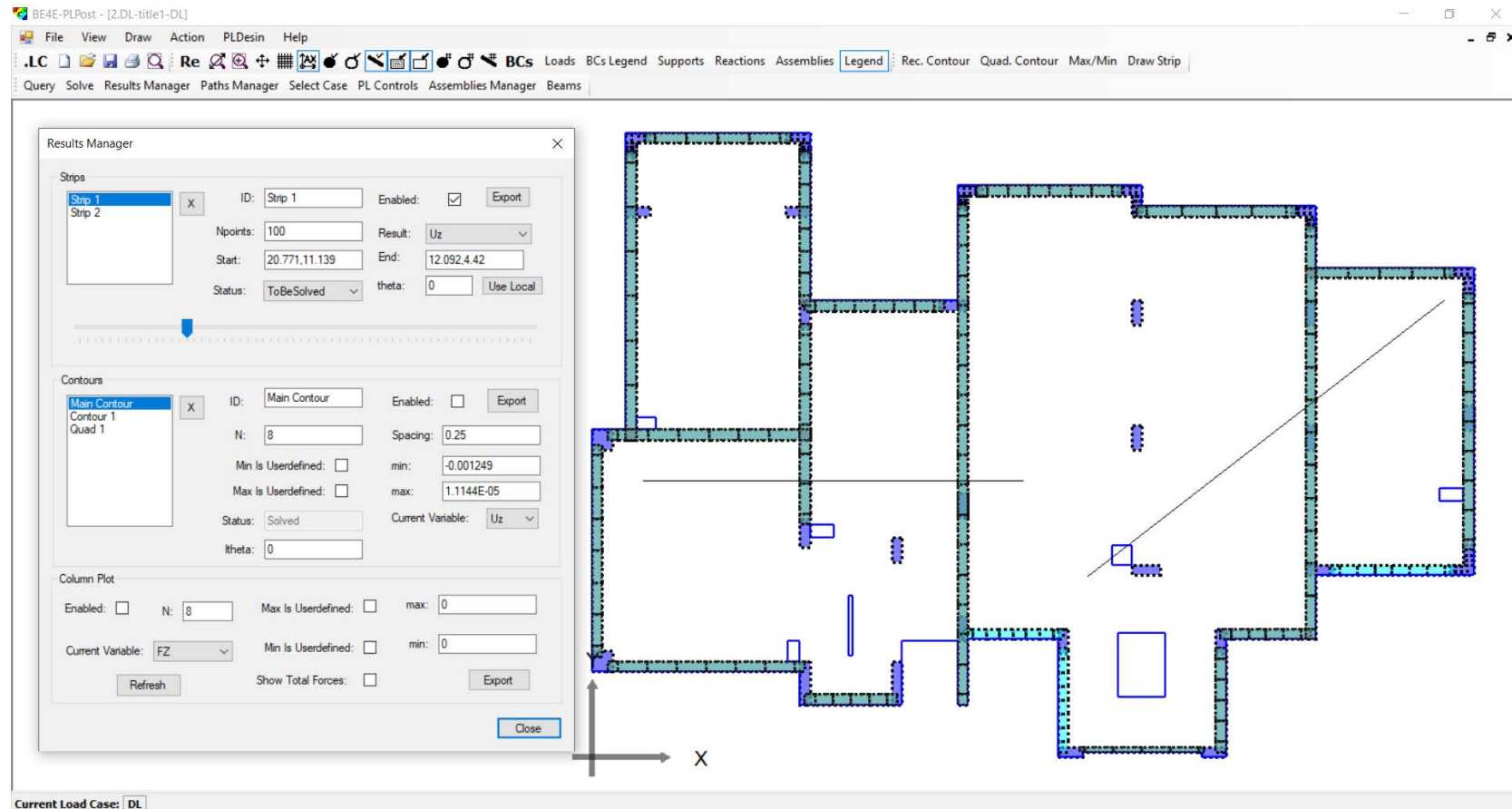
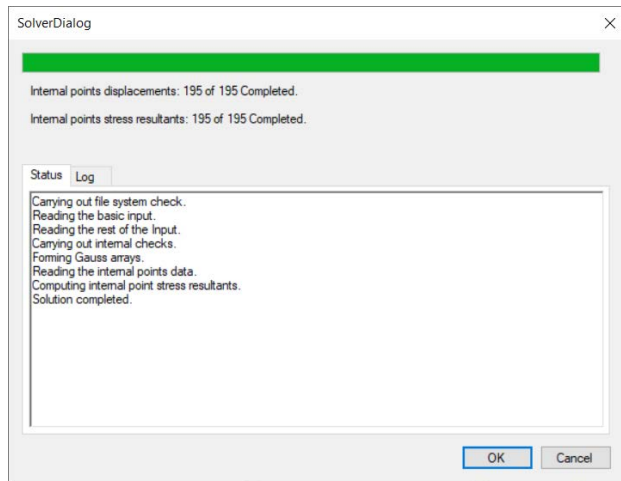
Loadcase	X	Y	Rx	Ry	Uz	Mxx
DL	2.87384	11.86531	-2.8171E-05	-0.00017283	-0.00074689	-0.29
LL	2.87384	11.86531	4.5301E-06	-8.4156E-05	-0.00045005	-0.19
Flooring	2.87384	11.86531	3.6241E-06	-6.7325E-05	-0.00036004	-0.15
Wall load	2.87384	11.86531	1.8121E-06	-3.3662E-05	-0.00018002	-0.07

Close

3. Slab results - strip result

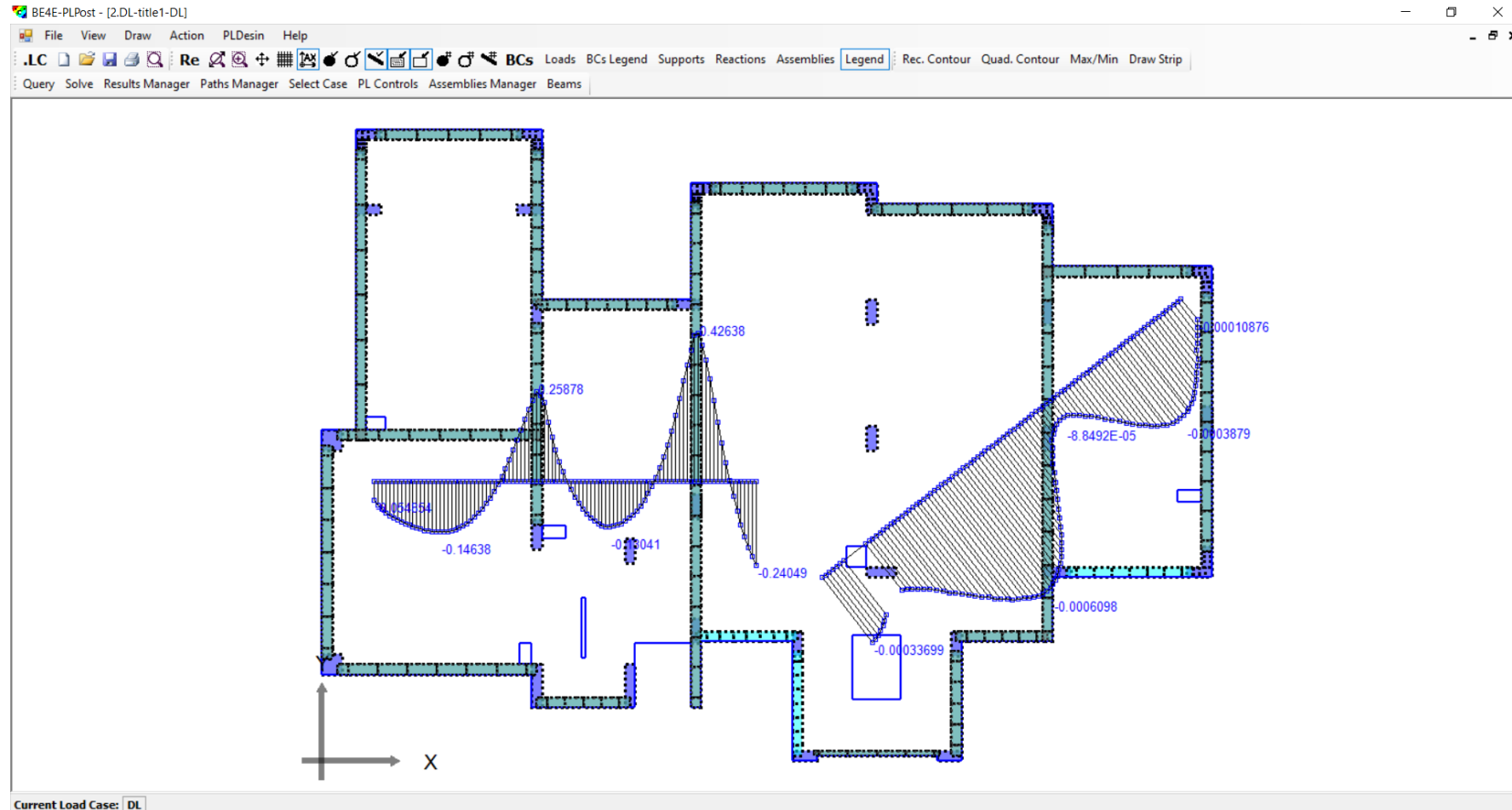
The user can see the straining action on strip:

- The user can draw inclined line by clicking the two points, or straight line by pressing shift during drawing.
- Then open the result manager to modify the strip.
- The user can insert number of points.
- The user can change the Start & the End points of the strip.
- Choose the result need to be shown in the strip.
- The user can export the result on text file after running the analysis.



3. Slab results - strip result

- After solving, press refresh or (F3) to show results.
- The user can change the Straining action need to be shown on strip by changing results in Result Manager.

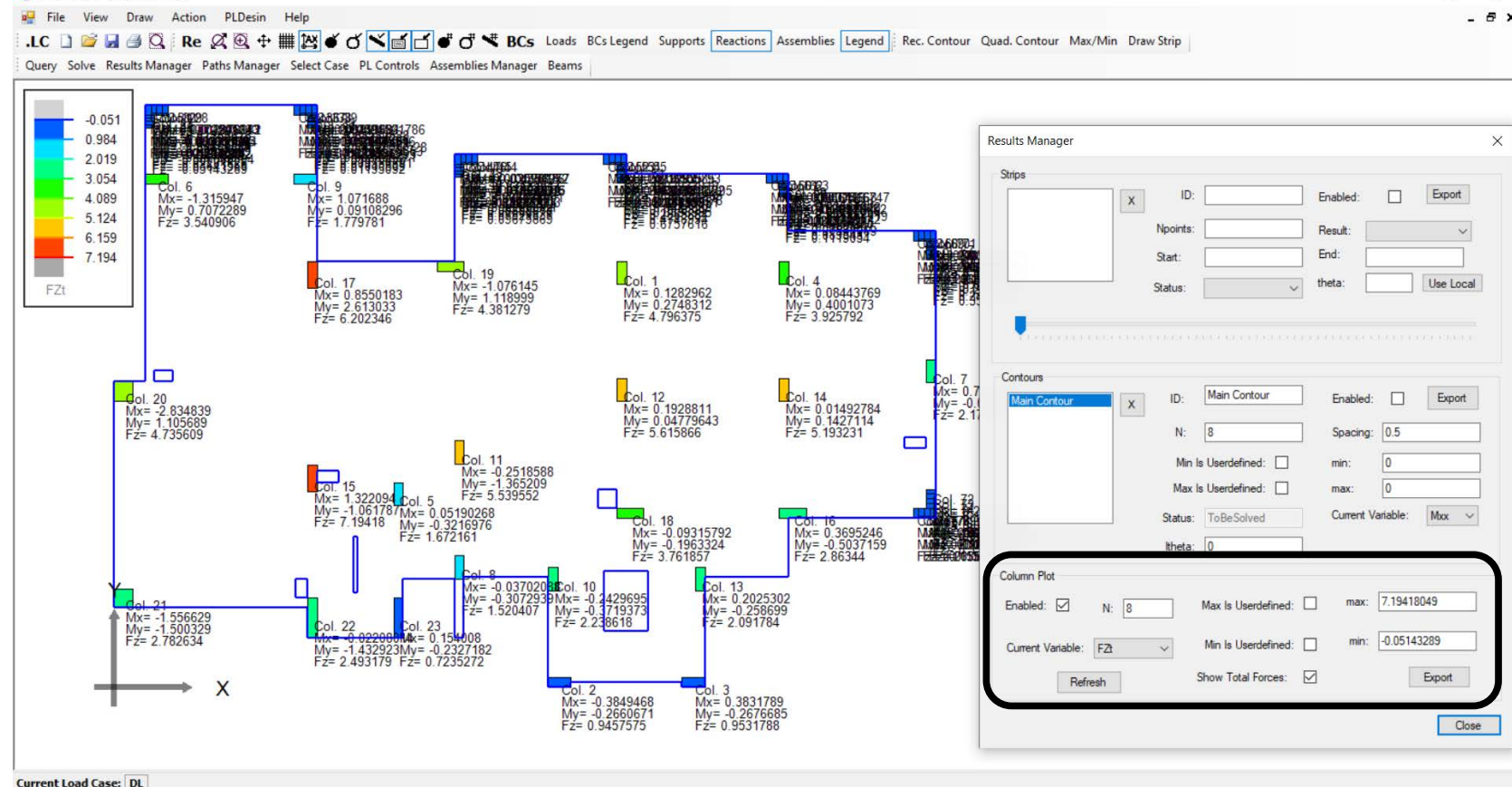


3. Slab results - supporting elements results

The user can show column analysis by two ways according to the type of model:

- In case of quadratic columns and without shear walls or beams.

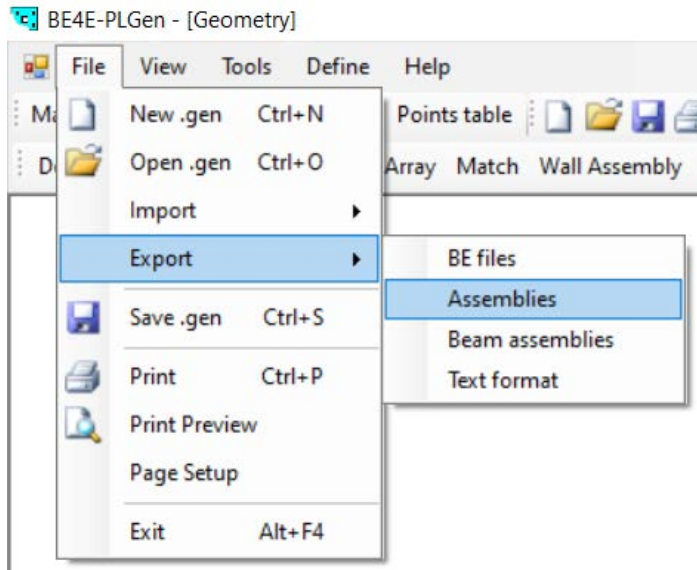
- Open the Result Manager, the last part is for column plot.
- The user mark on the check box (Enable).
- Mark on the check box (Show Total Force).
- Choose the Current Variable.
- The user can export the column straining actions on text file.
- Unlike slabs, the columns don't have to run.



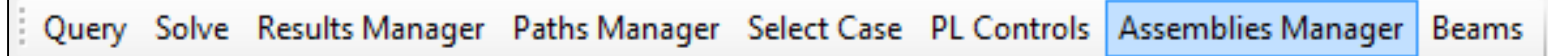
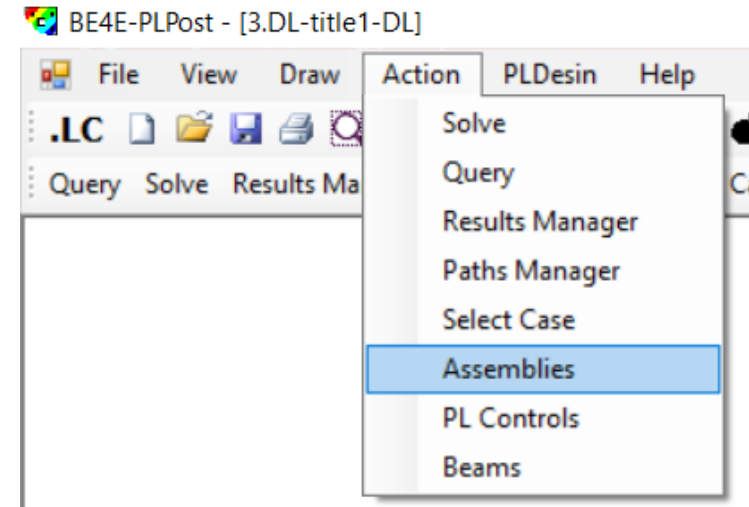
3. Slab results - supporting elements results

The user can show column analysis by two ways according to the type of model:

- In case of columns more than four sides or slab with shear walls or beams.
 - In this case the user should export Assembly file from generator file.
 - Then load the assembly file from Assemblies Manager tab.
 - From load tab select the assembly file (.asm) for loading all support elements calculating all geometric properties of the element.
 - Open the Result Manager then check mark on (Enable) box and check mark on (Show Total Force) box.



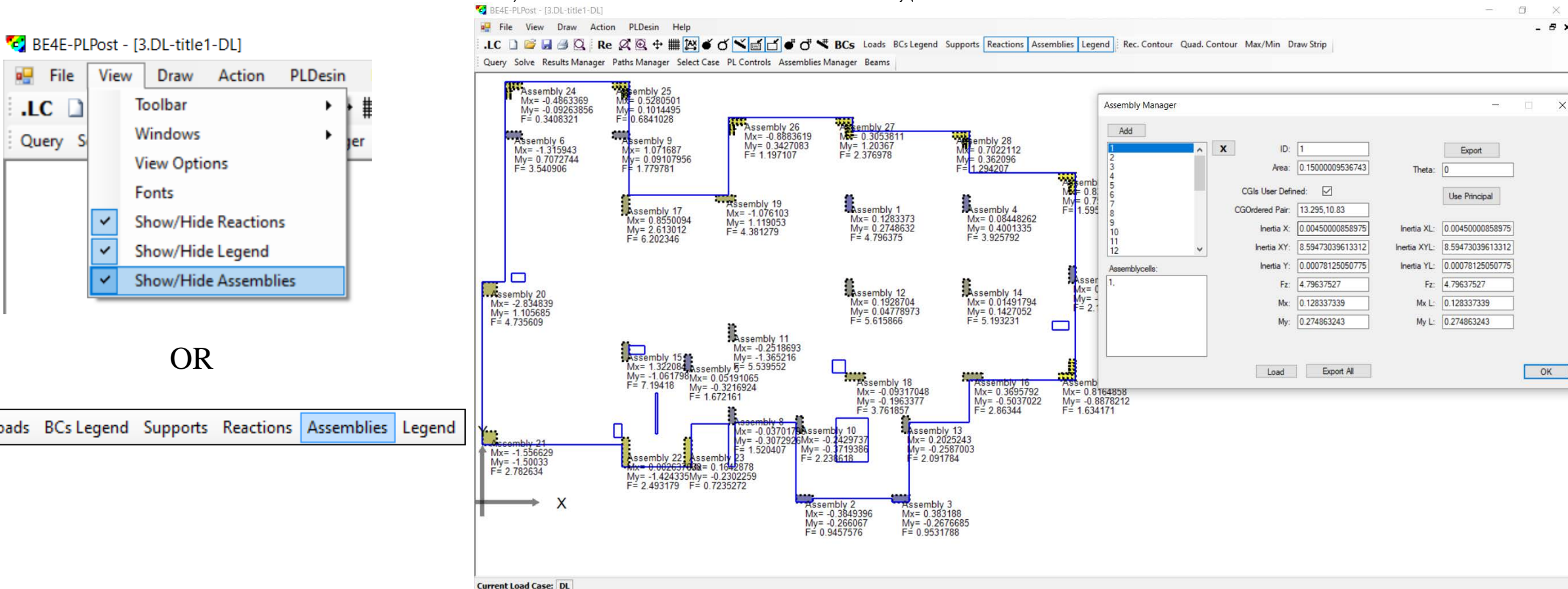
OR



3. Slab results - supporting elements results

The user can show column analysis by two ways according to the type of model:

- In case of columns more than four sides or slab with shear walls or beams.
- From Show/Hide Assemblies then refresh, the user can show the straining actions on columns or shear walls.



The screenshot displays the BE4E-PLPost software interface. The main window shows a structural model with various assemblies highlighted. The 'Assembly Manager' dialog box is open, showing the 'Add' tab with a list of assemblies and their properties. The 'Assembly Manager' dialog box includes the following fields:

ID:	1	Export	
Area:	0.1500009536743	Theta: 0	
CG User Defined:	<input checked="" type="checkbox"/>	Use Principal	
CGOrdered Pair:	13.295,10.83		
Inertia X:	0.00450000858975	Inertia XL:	0.00450000858975
Inertia XY:	8.59473039613312	Inertia XYL:	8.59473039613312
Inertia Y:	0.00078125050775	Inertia YL:	0.00078125050775
Fz:	4.79637527	Fz:	4.79637527
Mx:	0.128337339	Mx L:	0.128337339
My:	0.274863243	My L:	0.274863243

The main window also displays the following assembly results:

Assembly	Mx	My	F
Assembly 24	-0.4863369	-0.09263856	0.3408321
Assembly 25	0.5280501	0.1014495	0.6841028
Assembly 26	-0.8883619	0.3427083	1.197107
Assembly 27	0.3053811	1.20367	2.376978
Assembly 28	0.7022112	0.362096	1.294207
Assembly 6	-1.315943	0.7072744	3.540906
Assembly 9	1.071687	0.09107956	1.779781
Assembly 17	0.8550094	2.613012	6.202346
Assembly 19	-1.076103	1.119053	4.381279
Assembly 1	0.1283373	0.2748632	4.796375
Assembly 4	0.08448262	0.4001335	3.925792
Assembly 20	-2.834839	1.105685	4.735609
Assembly 11	-0.2518693	-1.365216	5.539552
Assembly 15	1.32208	0.05191065	7.19418
Assembly 18	-0.09317048	-0.1963377	3.761857
Assembly 16	0.3695792	-0.5037022	2.86344
Assembly 10	-0.3072928	-0.2429737	1.520407
Assembly 13	0.2025243	-0.2587003	2.091784
Assembly 22	-0.002637008	0.1642878	2.238618
Assembly 23	-1.424335	-0.2302259	2.493179
Assembly 2	-0.3849396	-0.266067	0.9457576
Assembly 3	0.383188	-0.2676685	0.9531788

The 'Assembly Manager' dialog box also includes the following fields:

CGs User Defined:	<input checked="" type="checkbox"/>
Inertia XL:	0.00450000858975
Inertia XYL:	8.59473039613312
Inertia YL:	0.00078125050775
Fz:	4.79637527
Mx L:	0.128337339
My L:	0.274863243

The 'Assembly Manager' dialog box also includes the following fields:

Load	Export All	OK
------	------------	----

The main window also displays the following assembly results:

Assembly	Mx	My	F
Assembly 24	-1.556629	-1.50033	2.782634
Assembly 23	-0.002637008	0.1642878	2.238618
Assembly 22	-1.424335	-0.2302259	2.493179
Assembly 21	0.0370178	-0.3072928	1.520407
Assembly 20	-2.834839	1.105685	4.735609
Assembly 19	-1.076103	1.119053	4.381279
Assembly 18	-0.09317048	-0.1963377	3.761857
Assembly 17	0.8550094	2.613012	6.202346
Assembly 16	0.3695792	-0.5037022	2.86344
Assembly 15	1.32208	0.05191065	7.19418
Assembly 14	0.01491794	0.1427052	5.615866
Assembly 13	0.2025243	-0.2587003	2.091784
Assembly 12	0.1928704	0.04778973	5.615866
Assembly 11	-0.2518693	-1.365216	5.539552
Assembly 10	-0.3072928	-0.2429737	1.520407
Assembly 9	1.071687	0.09107956	1.779781
Assembly 8	0.7022112	0.362096	1.294207
Assembly 7	0.3053811	1.20367	2.376978
Assembly 6	-1.315943	0.7072744	3.540906
Assembly 5	0.5280501	0.1014495	0.6841028
Assembly 4	0.08448262	0.4001335	3.925792
Assembly 3	0.383188	-0.2676685	0.9531788
Assembly 2	-0.3849396	-0.266067	0.9457576

The 'Assembly Manager' dialog box also includes the following fields:

Assemblycells:	1.
----------------	----

The 'Assembly Manager' dialog box also includes the following fields:

Export	Use Principal
--------	---------------

The 'Assembly Manager' dialog box also includes the following fields:

Load	Export All	OK
------	------------	----

The main window also displays the following assembly results:

Assembly	Mx	My	F
Assembly 24	-0.4863369	-0.09263856	0.3408321
Assembly 25	0.5280501	0.1014495	0.6841028
Assembly 26	-0.8883619	0.3427083	1.197107
Assembly 27	0.3053811	1.20367	2.376978
Assembly 28	0.7022112	0.362096	1.294207
Assembly 6	-1.315943	0.7072744	3.540906
Assembly 9	1.071687	0.09107956	1.779781
Assembly 17	0.8550094	2.613012	6.202346
Assembly 19	-1.076103	1.119053	4.381279
Assembly 1	0.1283373	0.2748632	4.796375
Assembly 4	0.08448262	0.4001335	3.925792
Assembly 20	-2.834839	1.105685	4.735609
Assembly 11	-0.2518693	-1.365216	5.539552
Assembly 15	1.32208	0.05191065	7.19418
Assembly 18	-0.09317048	-0.1963377	3.761857
Assembly 16	0.3695792	-0.5037022	2.86344
Assembly 10	-0.3072928	-0.2429737	1.520407
Assembly 13	0.2025243	-0.2587003	2.091784
Assembly 22	-0.002637008	0.1642878	2.238618
Assembly 23	-1.424335	-0.2302259	2.493179
Assembly 2	-0.3849396	-0.266067	0.9457576
Assembly 3	0.383188	-0.2676685	0.9531788

The 'Assembly Manager' dialog box also includes the following fields:

Load	Export All	OK
------	------------	----

The main window also displays the following assembly results:

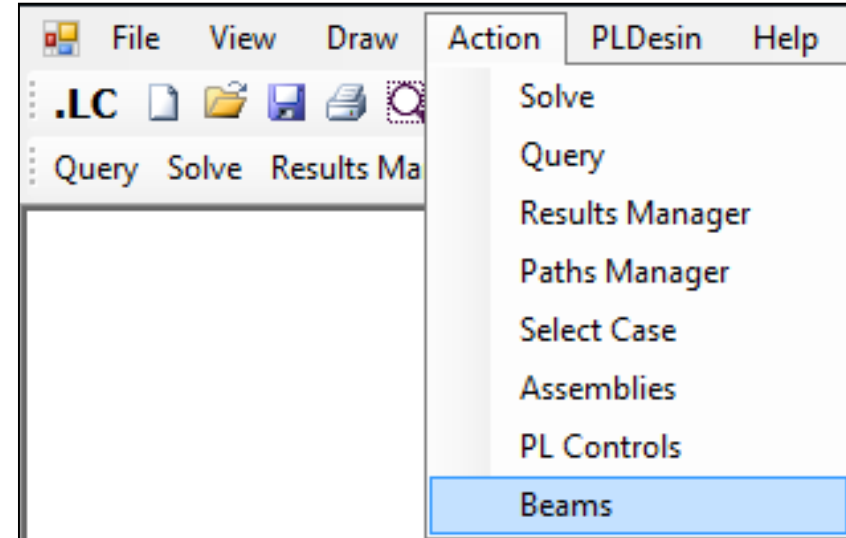
Assembly	Mx	My	F
Assembly 24	-1.556629	-1.50033	2.782634
Assembly 23	-0.002637008	0.1642878	2.238618
Assembly 22	-1.424335	-0.2302259	2.493179
Assembly 21	0.0370178	-0.3072928	1.520407
Assembly 20	-2.834839	1.105685	4.735609
Assembly 19	-1.076103	1.119053	4.381279
Assembly 18	-0.09317048	-0.1963377	3.761857
Assembly 17	0.8550094	2.613012	6.202346
Assembly 16	0.3695792	-0.5037022	2.86344
Assembly 15	1.32208	0.05191065	7.19418
Assembly 14	0.01491794	0.1427052	5.615866
Assembly 13	0.2025243	-0.2587003	2.091784
Assembly 12	0.1928704	0.04778973	5.615866
Assembly 11	-0.2518693	-1.365216	5.539552
Assembly 10	-0.3072928	-0.2429737	1.520407
Assembly 9	1.071687	0.09107956	1.779781
Assembly 8	0.7022112	0.362096	1.294207
Assembly 7	0.3053811	1.20367	2.376978
Assembly 6	-1.315943	0.7072744	3.540906
Assembly 5	0.5280501	0.1014495	0.6841028
Assembly 4	0.08448262	0.4001335	3.925792
Assembly 3	0.383188	-0.2676685	0.9531788
Assembly 2	-0.3849396	-0.266067	0.9457576

The 'Assembly Manager' dialog box also includes the following fields:

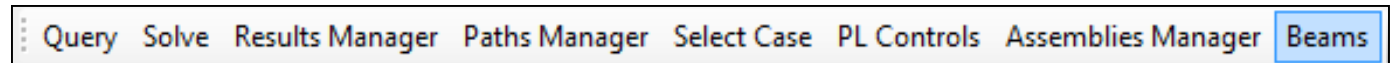
Load	Export All	OK
------	------------	----

4. Beam results

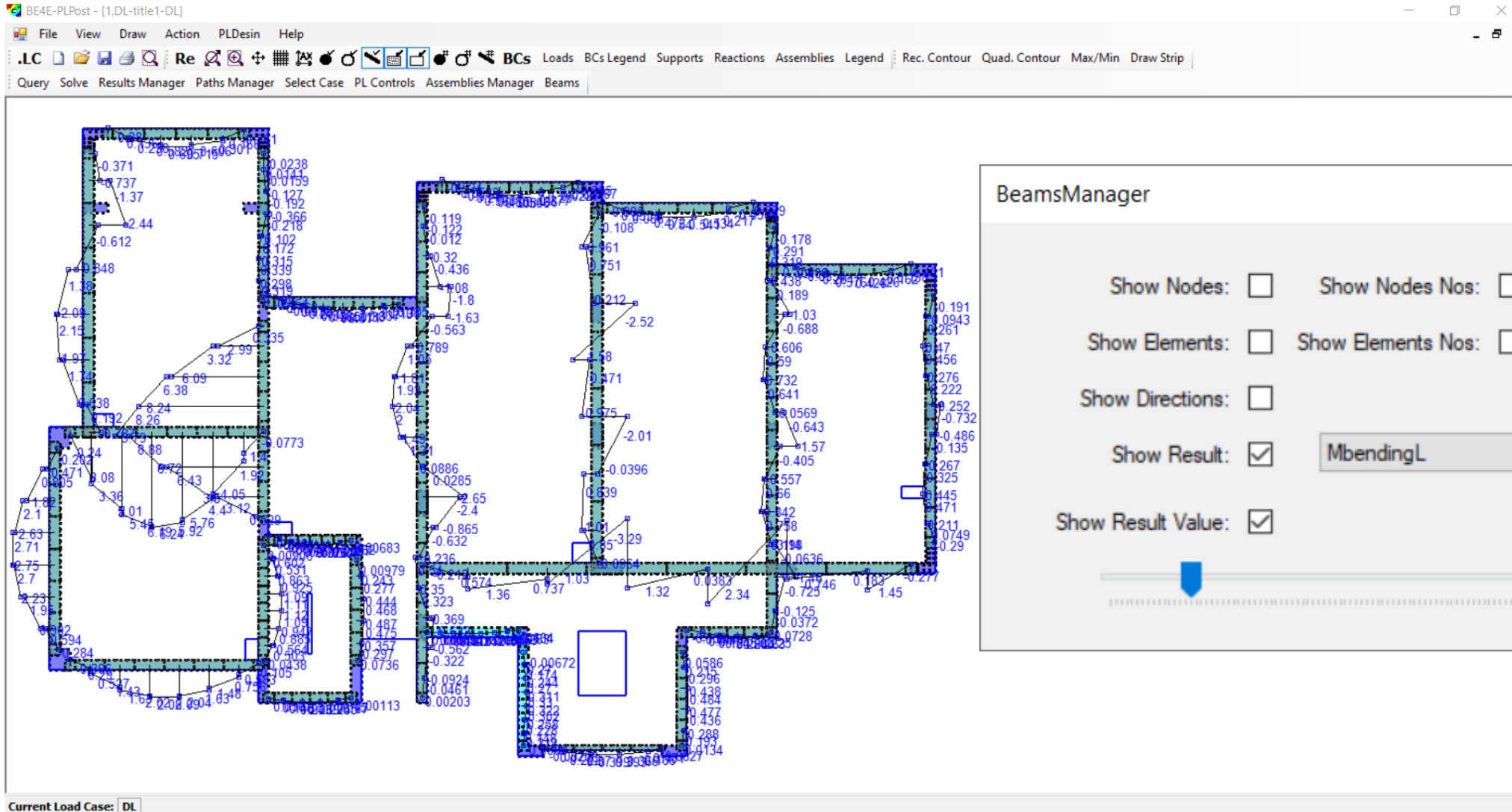
- Beams analysis is very simple just open Beams Manager tab.
- Press on Read Beam Geometry.
- Press on Read Beams Results.
- Check mark on Show Result box.
- Check mark on Result Value box.
- Choose the Straining action need to be shown.
- Like column analysis, beams don't need to run.



OR



4. Beam results



BeamsManager

Show Nodes: Show Nodes Nos:

Show Elements: Show Elements Nos:

Show Directions:

Show Result: MbendingL

Show Result Value:

Read Beam Geometry

Read Default Beam Results

Browse Required Beam Results

Export

OK

CUFE-BE



www.be4e.com



Cairo University

- A. PLGen – Model generator** ✓
- B. PLView – Numerical model** ✓
- C. PLCoreMan – Manager and solver** ✓
- D. PLPost – Post processing** ✓
- E. PLDesign – Design tool**
- F. PLPAK modelling capabilities**

CUFE-BE



www.be4e.com



Cairo University

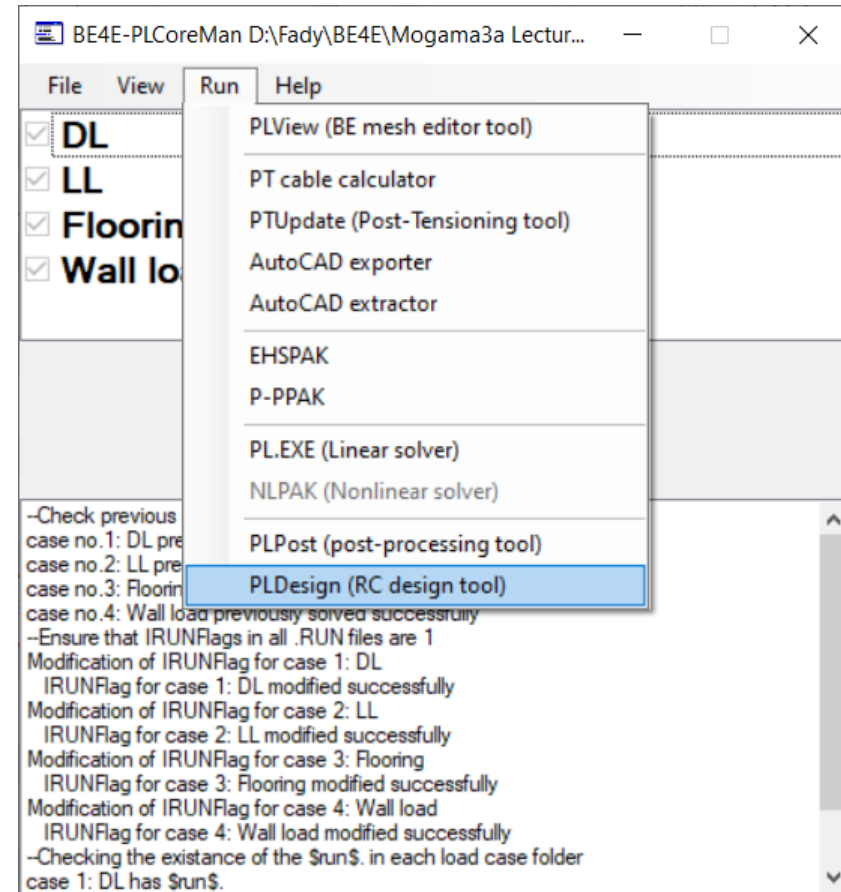
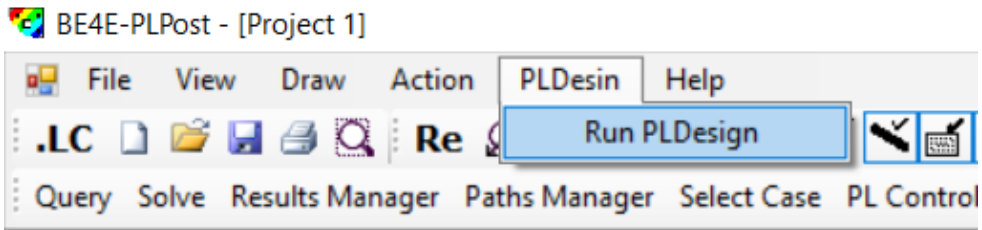
- A. PLGen – Model generator
- B. PLView – Numerical model
- C. PLCoreMan – Manager and solver
- D. PLPost – Post processing
- E. PLDesign – Design tool**
- F. PLPAK modelling capabilities

What is the PLDesign?

- PLDesign (Plate design package) is a structural design tool package for plate bending structures based on the boundary element method for shear deformable plate bending theory, using different codes like (ACI, EC, and ECP).
- The PLDesign is added to the PLPAK-Basic package to design reinforced concrete building slabs and foundations.
- The PLDesign is not only consider about design, but also about detailing and calculation sheet forming.
- In the PLDesign the user can check the reinforcement of section under any stresses (Bending, Shear and Torsion).
- In the PLDesign the user can check deflection and punching of slabs.
- In the PLDesign the calculation is not only for load combination, but also for envelopes.

What is the PLDesign?

- The user can go to PLDesign by two ways either by using PLPost or PLCoreMan as follows:



CUFE-BE



www.be4e.com



Cairo University

PLDesign Package

2.1. File needed to be exported before using PLDesign

2.2. Starting PLDesign

2.3. Load combinations & load envelopes

2.4. Slab design

2.4.1. Design from PLPost results (strip design)

2.4.2. Design from PLPost results (contour design)

2.4.3. Design from PLDesign directly (strip based region)

2.4.4 Design from PLDesign directly (basic and additional reinforcement)

2.5 Check deflections of slab

2.6 Check punching

2.7 Beam design

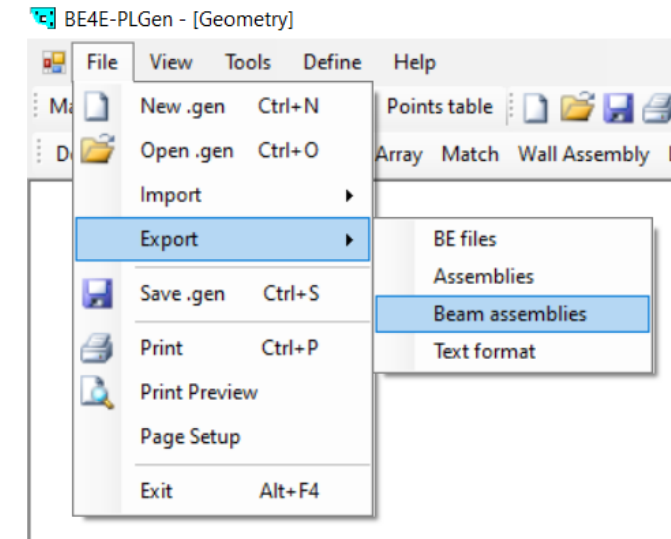
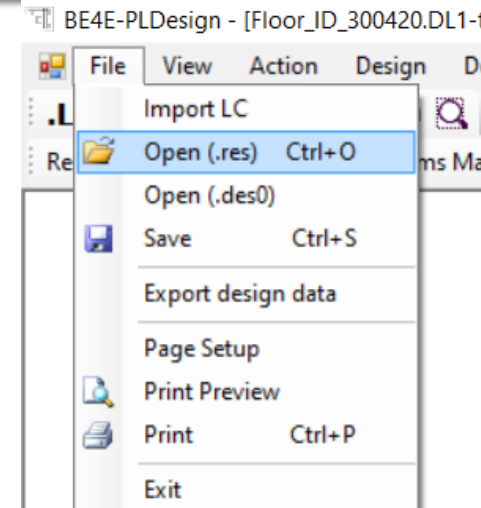
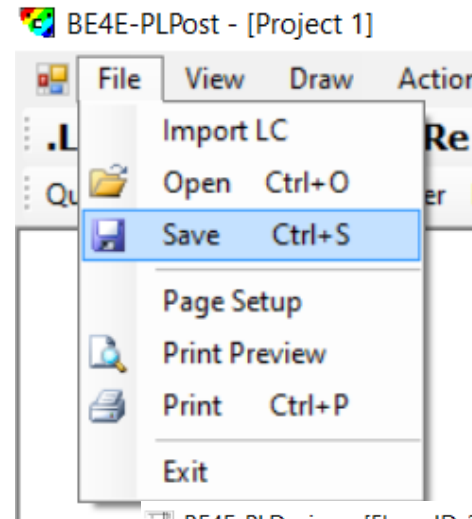
1. File needed to be exported before using PLDesign

There are cases that user have to export file from PLGen and PLPost before using PLDesign:

- Export beam assemblies: this case is used to design the beams.
- Save the PLPost results: this case is used to design the slab.
- Export assemblies file: this case is used to check punching of the columns.

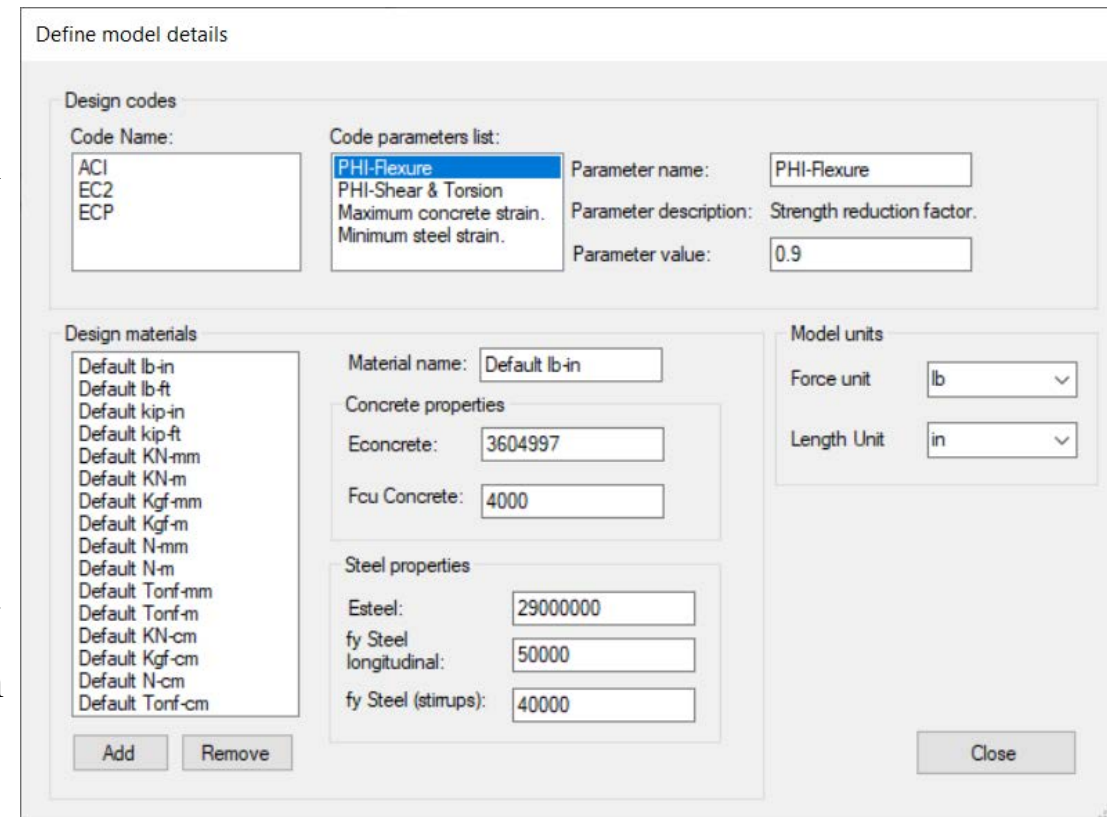
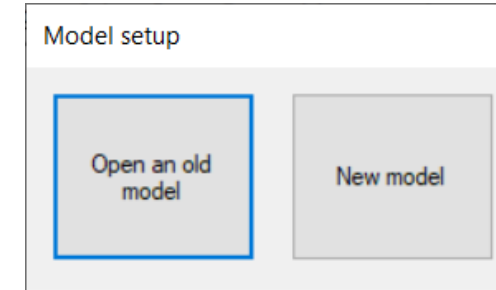
The previous cases can be restored in the PLDesign as follows:

- Import beam assemblies: this case is used to design the beams (will be demonstrated in beam design section).
- Open the PLPost results: this case is used to design the slab.
- Import assemblies file: this case is used to check punching of the columns.



2. Starting PLDesign

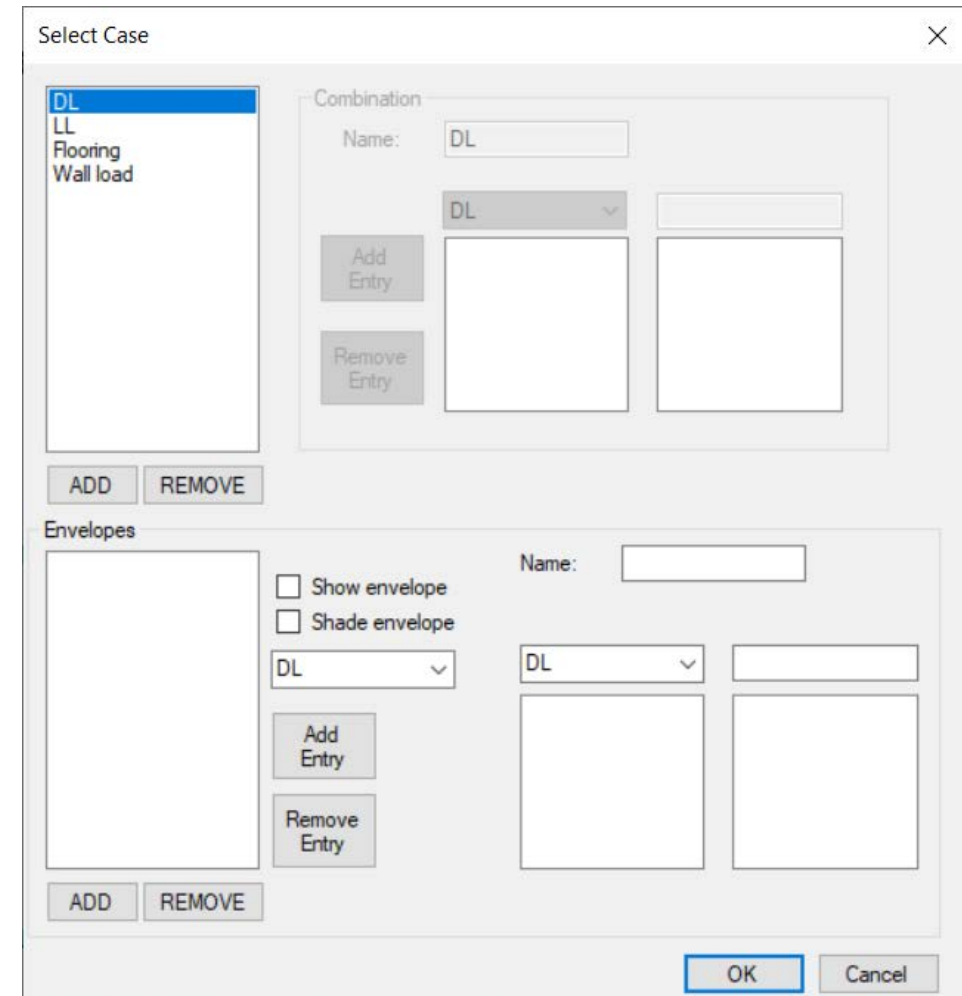
- Once the user opens the PLDesign, a model setup window is opened and asks if the model is a new model or an old model.
- As it starts in PLDesign we will click on New model.
- In case of using PLDesign before, the user can click on open an old model.
- After clicking on New model, the user should define model details.
- The model details are the code name and the code parameter, design material units and modify it if any, choosing model units.
- To save click close then choose (.LC) file need to be designed, it has to be noted that it will be opened automatically if opened from PLCoreMan.



3. Load combinations & load envelopes

- The lower tabs of the PLDesign contain by default current load case is dead and current envelope is none.
- If the user press double click on dead load combinations window will open.
- The user can add cases like ultimate, working cases, each case contains combination between load cases inserted from PLGen and if there are any envelopes between them the user can insert also the envelope between combinations to achieve max. straining actions.

Current Load Case: **DL** Current Load Envelope: **None**



Select Case

DL
LL
Flooring
Wall load

Combination

Name: DL

DL

Add Entry

Remove Entry

ADD REMOVE

Envelopes

Show envelope
Shade envelope

Name:

DL

DL

Add Entry

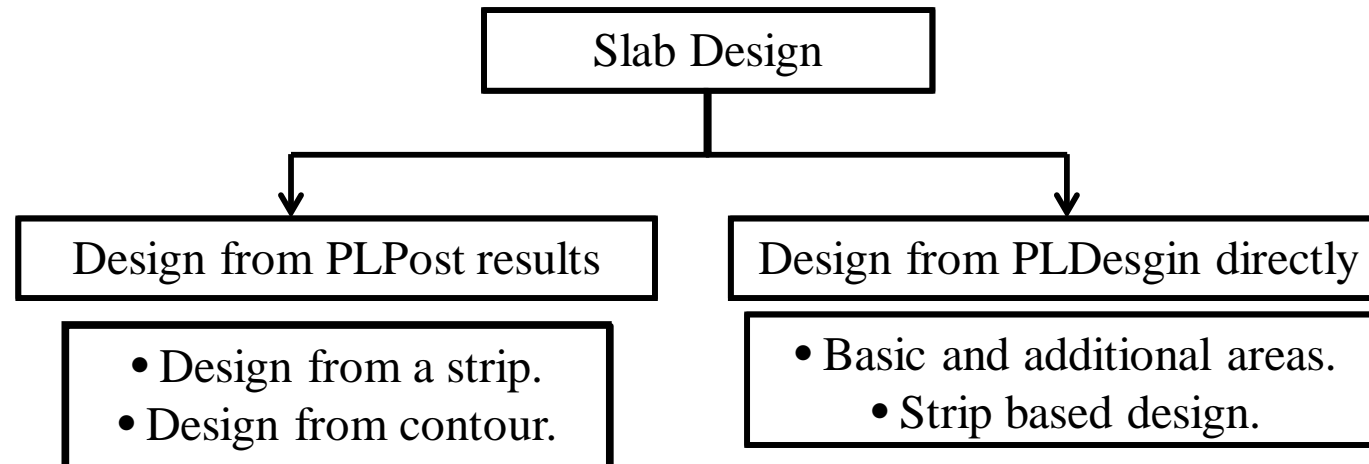
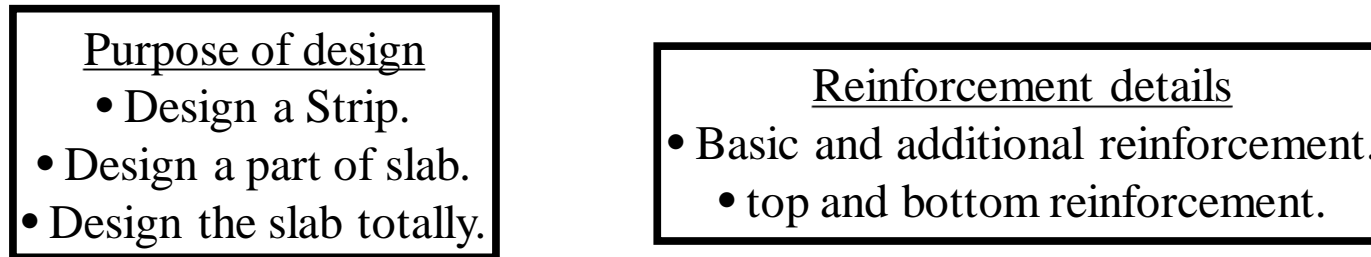
Remove Entry

ADD REMOVE

OK Cancel

4. Slab design

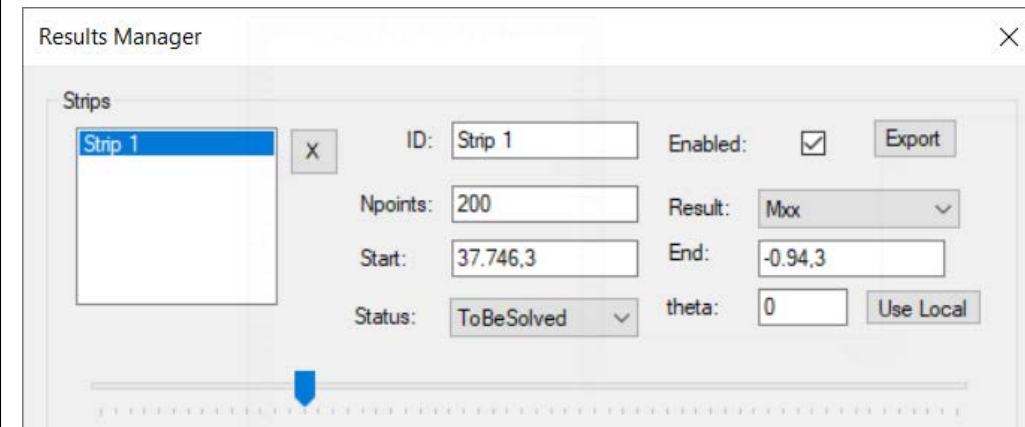
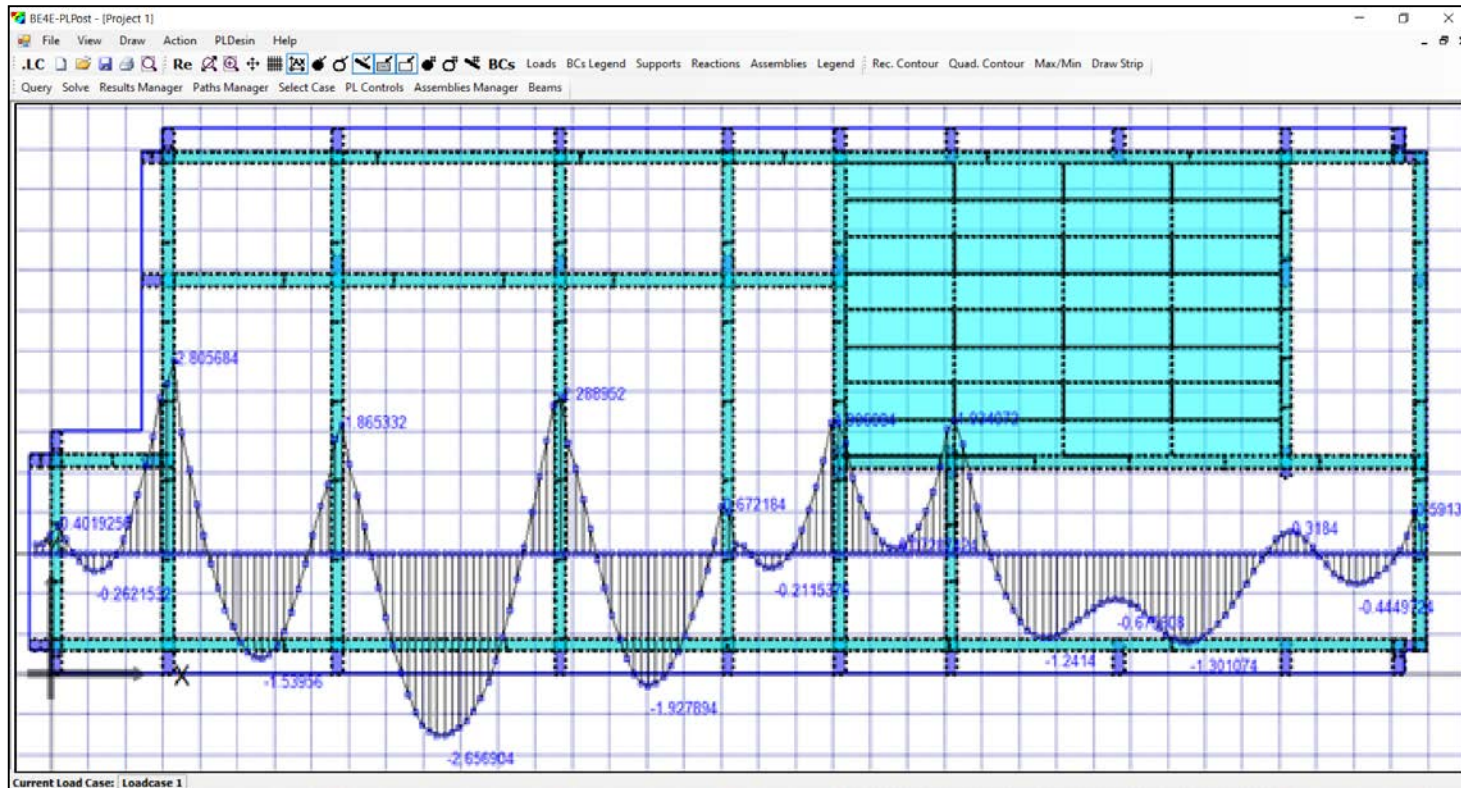
The user have many options to design slabs, these options are according to the purpose of design or what are the details that user need from PLDesign:



4.1. Design from PLPost results (strip design)

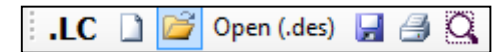
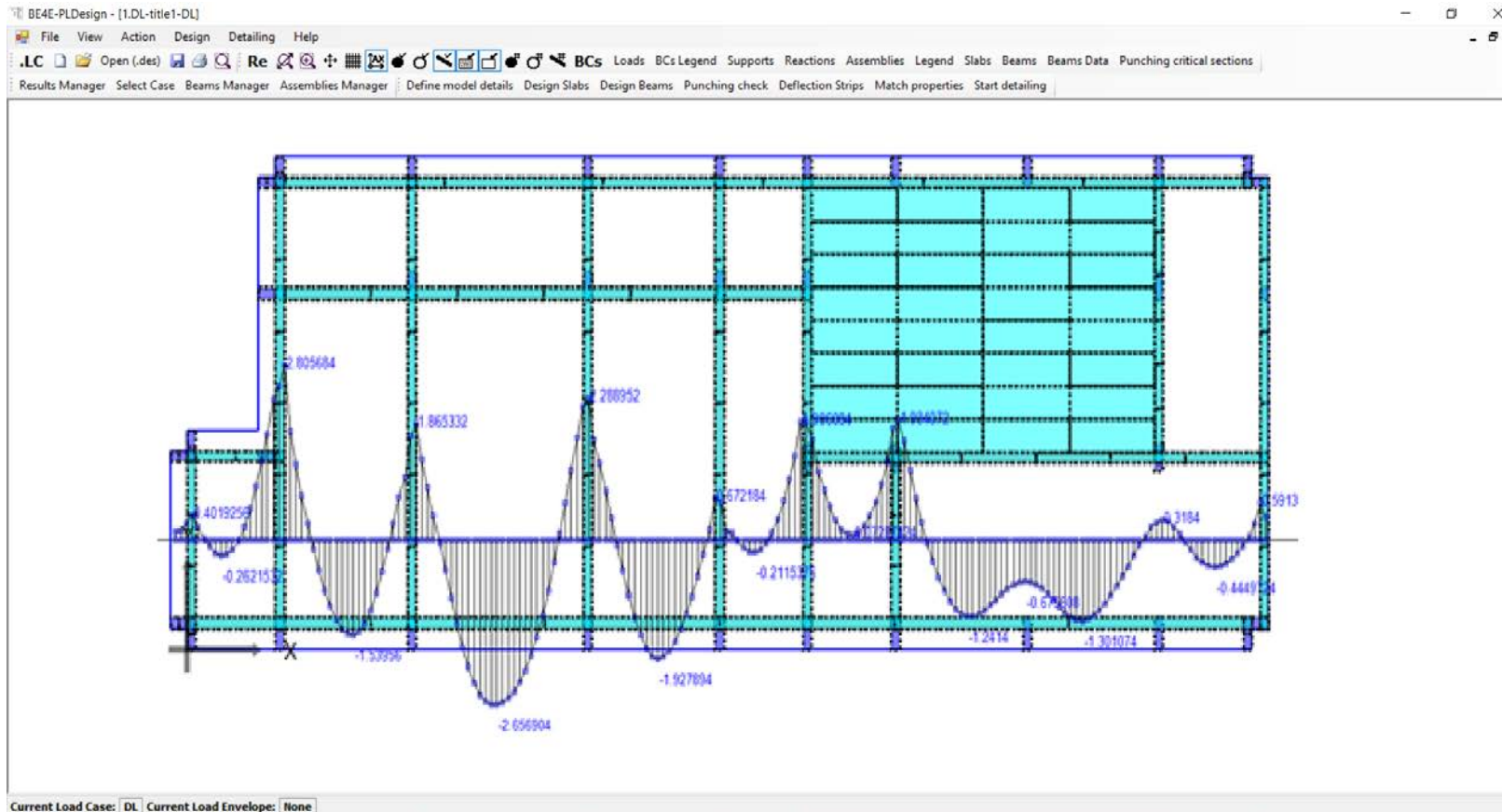
Design a certain strip with specific coordinates is one of the advantages of the PLDesign.

First the user have to save a strip from PLPost (a strip at $y = 3.00\text{m}$) as (.res) file.

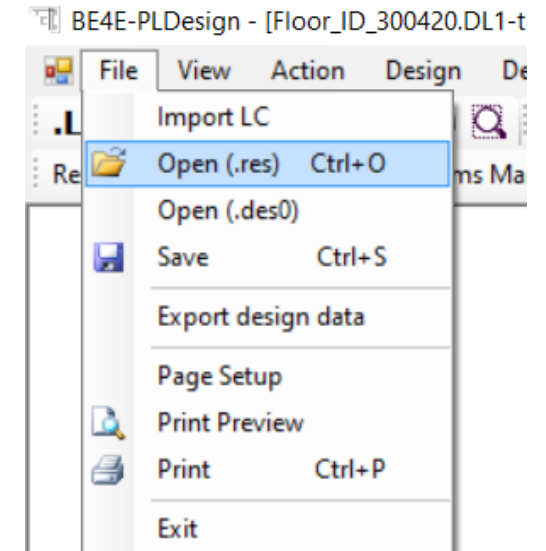


4.1. Design from PLPost results (strip design)

- Load the saved (.res) file in PLDesign.



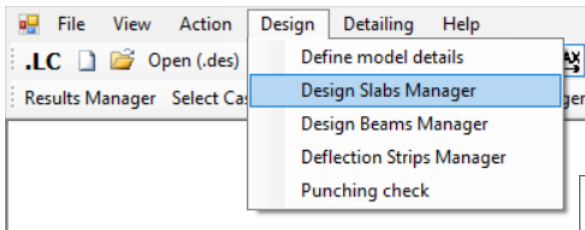
OR



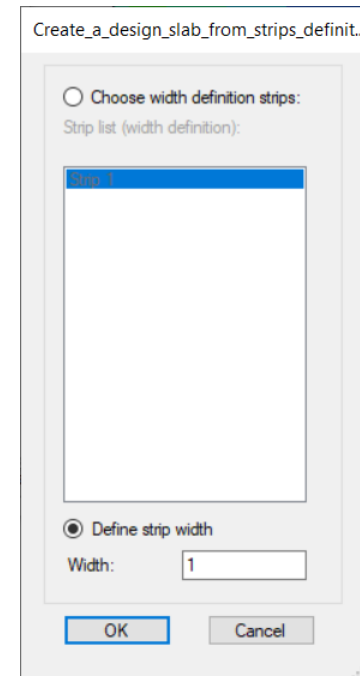
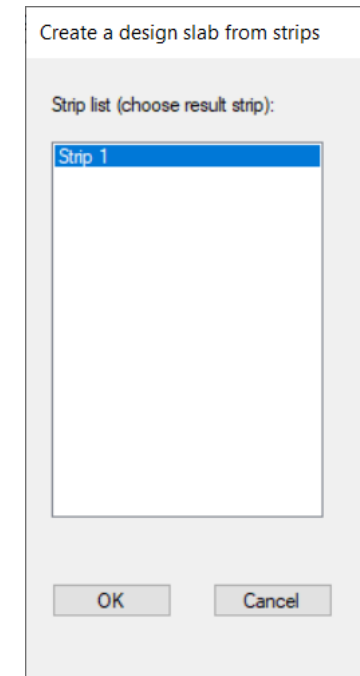
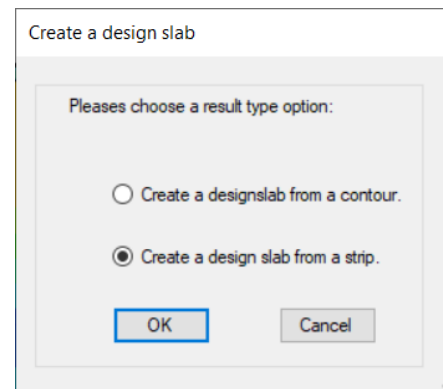
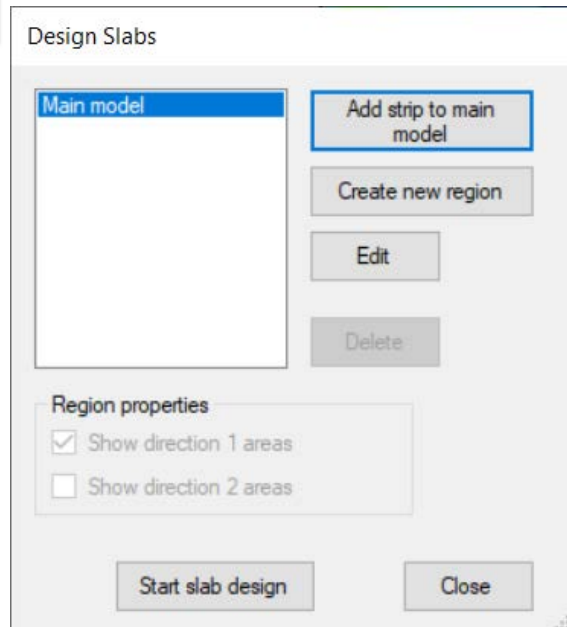
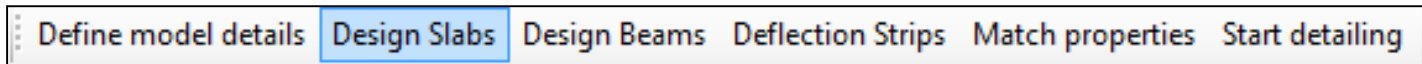
4.1. Design from PLPost results (strip design)

- Open Design Slabs Manager, then press on add strip to main model tab.
- Choose a design slab from a strip which will open a strip list to choose from them, then determine the width of the strip.

BE4E-PLDesign - [Project 2]

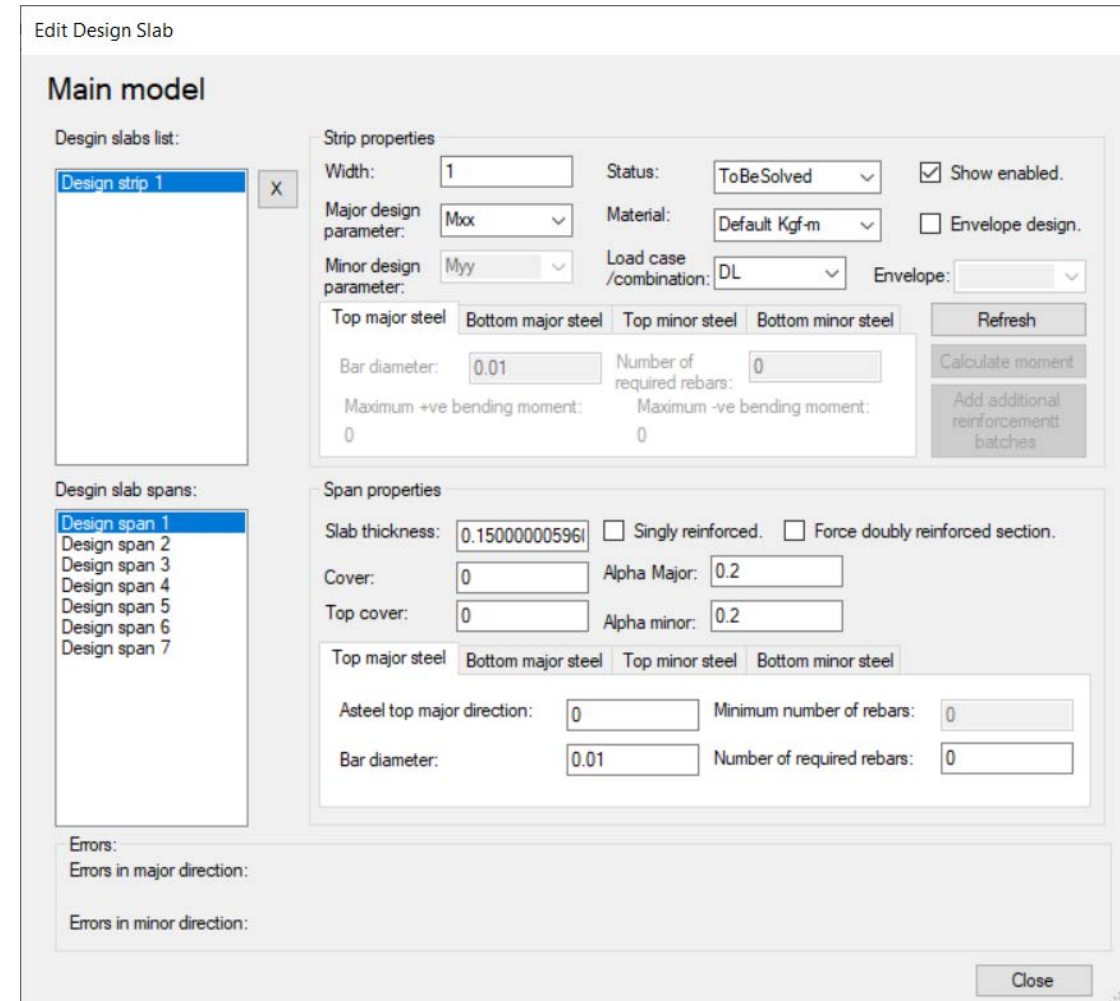
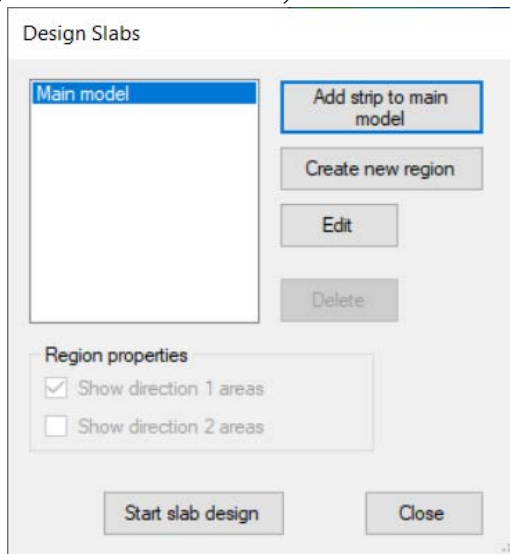


OR



4.1. Design from PLPost results (strip design)

- Press on Edit to open Design Slab Manager.
- The main model is divided into two parts: Strip properties and Span properties.
- The strip properties contains information for the strip like width, Major design parameter, Material units, Load cases/combinations.
- While span properties contains information for the section like thickness, cover, bar diameter, number of rebars, number of required rebars.



4.1. Design from PLPost results (strip design)

- Adjust the main model by choosing the design parameter and inserting all information (cover, bar diameter and number of bars) for one span.
- User do not need to adjust all spans, user can match the spans with the same properties.

Define model details Design Slabs Design Beams Deflection Strips **Match properties** Start detailing

Edit Design Slab

Main model

Design slabs list:

- Design strip 1

Strip properties

Width: 1 Status: ToBeSolved Show enabled.

Major design parameter: Mox Material: Default Kgf-m Envelope design.

Minor design parameter: Myy Load case /combination: DL Envelope:

Top major steel Bottom major steel Top minor steel Bottom minor steel

Bar diameter: 0.01 Number of required rebars: 0

Maximum +ve bending moment: 0 Maximum -ve bending moment: 0

Design slab spans:

- Design span 1
- Design span 2
- Design span 3
- Design span 4
- Design span 5
- Design span 6
- Design span 7

Span properties

Slab thickness: 0.15000000596 Singly reinforced. Force doubly reinforced section.

Cover: 0 Alpha Major: 0.2

Top cover: 0 Alpha minor: 0.2

Top major steel Bottom major steel Top minor steel Bottom minor steel

Asteel top major direction: 0 Minimum number of rebars: 0

Bar diameter: 0.01 Number of required rebars: 0

Errors:

Errors in major direction:

Errors in minor direction:

Match properties

Slab spans Beams Beam sections Punching asms.

Source region: Main model Destination region: Main model

Source area: Design strip 1 Destination area: Design strip 1

Source span: Design span 1 Design span 2 Design span 3 Design span 4 Design span 5 Design span 6 Design span 7

Destination span: Design span 1 Design span 2 Design span 3 Design span 4 Design span 5 Design span 6 Design span 7

Top major steel

- Bar diameters
- Bar amounts

Top minor steel

- Bar diameters
- Number of bars

Bottom major steel

- Bar diameters
- Number of bars

Bottom minor steel

- Bar diameters
- Number of bars

Dimensions

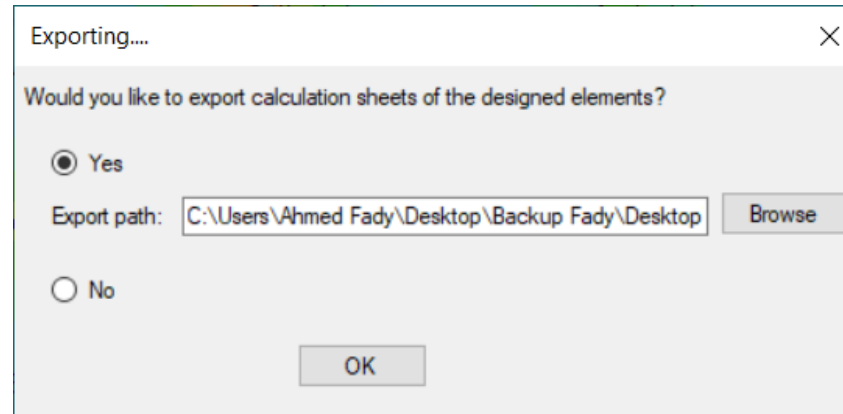
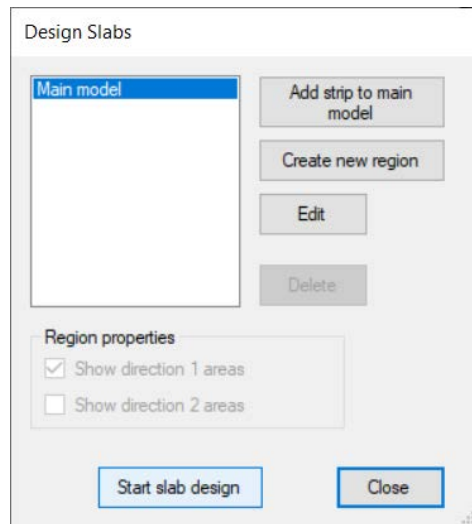
- Slab thickness
- Bottom cover
- Top cover

Section data

- Is Singly reinforced
- Force doubly reinforced section.
- Alpha values

4.1. Design from PLPost results (strip design)

- Check that all spans are matched then start slab design.
- The PLDesign is automatically export the calculation sheet for slab every section has two files one for moment in X-direction and the other for Y-direction.



PLDESIGN : Design Calculation Sheet

FLEXURAL DESIGN of a Single Reinforced Rectangular Section According to ECP

Slab No: Design strip 1:Design span 9
Region ID: Main model type of section: Slab

Company Name: _____
Project Name: _____
Designed By: _____
Reviewed By: _____
Approved by: _____

Dimensions & Moment		Materials	
Moment (M)	1E+08 N.mm	Steel yield Strength (fy)	353.039 N/mm ²
Thickness of section (t)	200 mm	Concrete Cube Strength (fcu)	24.5168 N/mm ²
Concrete clear cover ϕ	10 mm	Steel Young's Modulus (E)	205940 N/mm ²
Depth of Section (d)	190 mm	Concrete Strain (ϵ)	0.003
		Partial Factors	γ_c 1.5 γ_s 1.15

Design

$$a = \frac{(0.67 \cdot f_{cu} \cdot b \cdot d) - \sqrt{[(0.67 \cdot f_{cu} \cdot b \cdot d) - 4 \cdot (0.67 \cdot f_{cu} \cdot b \cdot d) \cdot (1.5 \cdot M/2)]}}{2 \cdot (0.67 \cdot f_{cu} \cdot b)}$$

a= 70.50904436 mm

$$c_{max} = \frac{2}{3} \cdot \frac{E_{cmax}}{E_{cmax} + \frac{f_y}{\gamma_s} / E_{steel}}$$

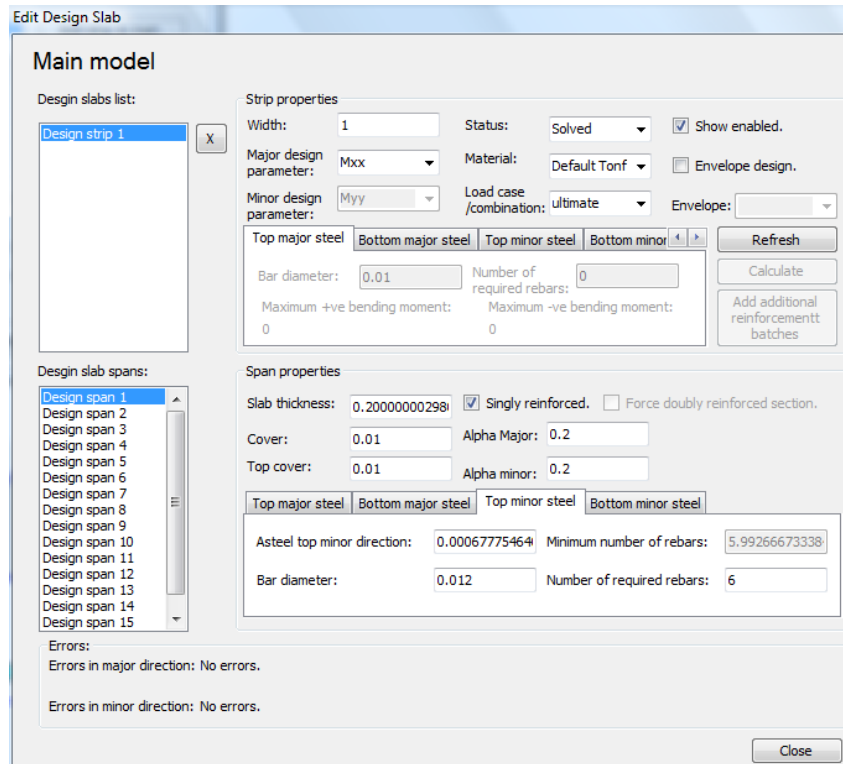
Cmax= 0.445368528 mm

$c = \frac{a}{0.8/d}$ **Check C<Cmai**

Page 1

4.1. Design from PLPost results (strip design)

- Check that all spans are safe and modify the number of bars and bar diameter if needed.



Edit Design Slab

Main model

Design slabs list:

- Design strip 1

Strip properties

Width: 1 Status: Solved Show enabled.

Major design parameter: Mxx Material: Default Tonf Envelope design.

Minor design parameter: Myy Load case /combination: ultimate Envelope:

Top major steel Bottom major steel Top minor steel Bottom minor

Refresh

Bar diameter: 0.01 Number of required rebars: 0

Calculate

Maximum +ve bending moment: 0 Maximum -ve bending moment: 0

Add additional reinforcement batches

Design slab spans:

- Design span 1
- Design span 2
- Design span 3
- Design span 4
- Design span 5
- Design span 6
- Design span 7
- Design span 8
- Design span 9
- Design span 10
- Design span 11
- Design span 12
- Design span 13
- Design span 14
- Design span 15

Span properties

Slab thickness: 0.2000000298 Singly reinforced. Force doubly reinforced section.

Cover: 0.01 Alpha Major: 0.2

Top cover: 0.01 Alpha minor: 0.2

Top major steel Bottom major steel Top minor steel Bottom minor steel

Asteel top minor direction: 0.00067775464 Minimum number of rebars: 5.99266673338

Bar diameter: 0.012 Number of required rebars: 6

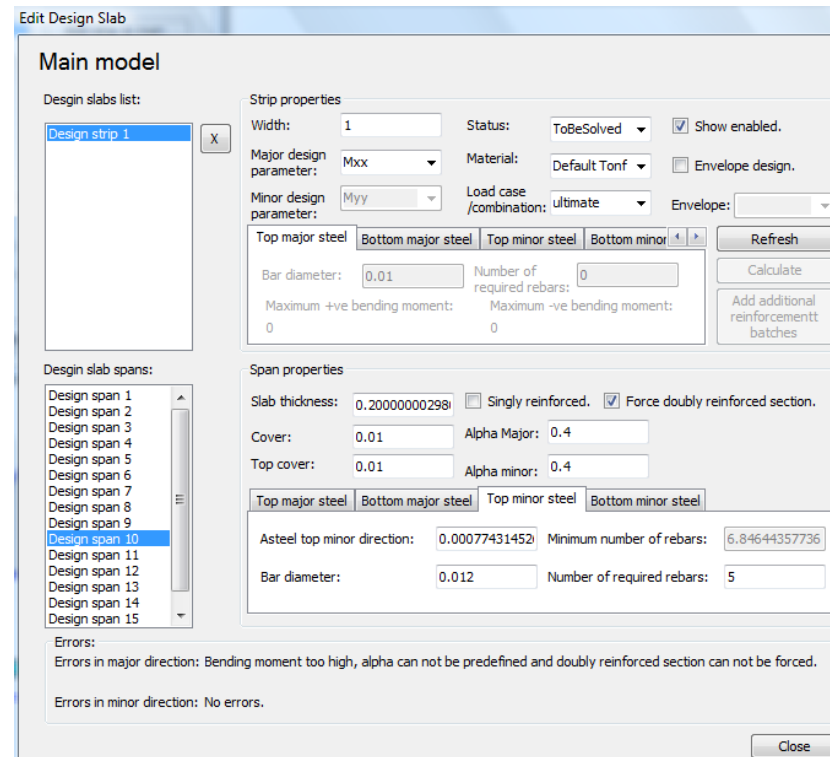
Errors:

Errors in major direction: No errors.

Errors in minor direction: No errors.

Close

Case of safe section



Edit Design Slab

Main model

Design slabs list:

- Design strip 1

Strip properties

Width: 1 Status: ToBeSolved Show enabled.

Major design parameter: Mxx Material: Default Tonf Envelope design.

Minor design parameter: Myy Load case /combination: ultimate Envelope:

Top major steel Bottom major steel Top minor steel Bottom minor

Refresh

Bar diameter: 0.01 Number of required rebars: 0

Calculate

Maximum +ve bending moment: 0 Maximum -ve bending moment: 0

Add additional reinforcement batches

Design slab spans:

- Design span 1
- Design span 2
- Design span 3
- Design span 4
- Design span 5
- Design span 6
- Design span 7
- Design span 8
- Design span 9
- Design span 10
- Design span 11
- Design span 12
- Design span 13
- Design span 14
- Design span 15

Span properties

Slab thickness: 0.2000000298 Singly reinforced. Force doubly reinforced section.

Cover: 0.01 Alpha Major: 0.4

Top cover: 0.01 Alpha minor: 0.4

Top major steel Bottom major steel Top minor steel Bottom minor steel

Asteel top minor direction: 0.00077431452 Minimum number of rebars: 6.84644357736

Bar diameter: 0.012 Number of required rebars: 5

Errors:

Errors in major direction: Bending moment too high, alpha can not be predefined and doubly reinforced section can not be forced.

Errors in minor direction: No errors.

Close

Case of unsafe section

4.1. Design from PLPost results (strip design)

- Instead of every strip contains two excel files, the user can export a summary for slab reinforcement.

Export design data

Export slabs

Slab regions:

Main model

Select all Deselect all

Export beams

Design beams:

Select all Deselect all

Export punching assemblies

Punching assemblies:

Select all Deselect all

Export reinforcement to Revit

Level name in Revit:

Export

Export summary files (.xls)

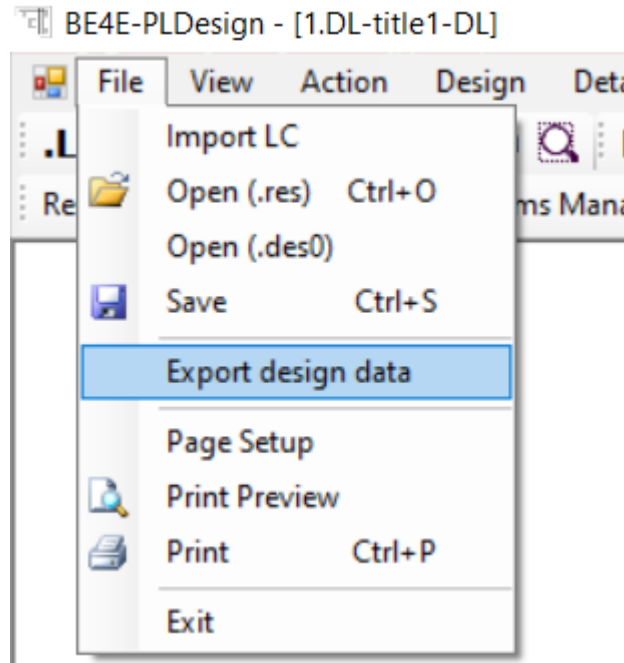
Export beams

Export slabs

Export punching assemblies

Export

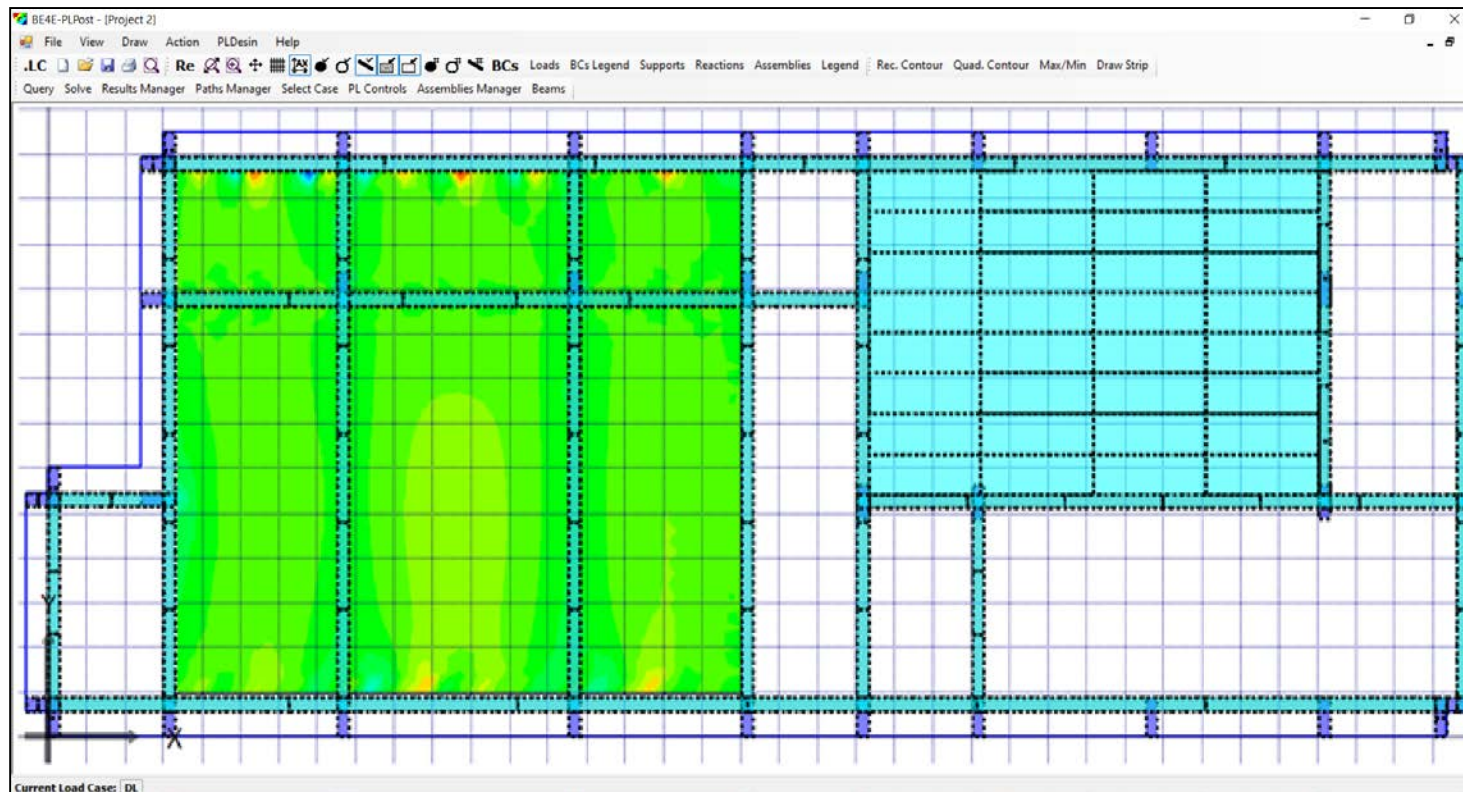
Close



Region name	Area name	Major design moment	Strip name	Top major rft.	Bot. major rft.	Top minor rft.	Bot minor rft.
Main model	Design strip 1	Mxx	Design span 1	5 ϕ 0.012	6 ϕ 0.012	6 ϕ 0.012	5 ϕ 0.012
			Design span 2	5 ϕ 0.012	5 ϕ 0.012	10 ϕ 0.016	5 ϕ 0.012
			Design span 3	5 ϕ 0.012	6 ϕ 0.012	10 ϕ 0.016	5 ϕ 0.012
			Design span 4	6 ϕ 0.016	5 ϕ 0.012	9 ϕ 0.018	5 ϕ 0.012
			Design span 5	5 ϕ 0.012	9 ϕ 0.018	5 ϕ 0.012	7 ϕ 0.012
			Design span 6	6 ϕ 0.012	5 ϕ 0.012	6 ϕ 0.012	5 ϕ 0.012
			Design span 7	5 ϕ 0.012	7 ϕ 0.012	7 ϕ 0.012	5 ϕ 0.012
			Design span 8	9 ϕ 0.016	5 ϕ 0.012	7 ϕ 0.016	5 ϕ 0.012
			Design span 9	5 ϕ 0.012	5 ϕ 0.012	5 ϕ 0.012	5 ϕ 0.012
			Design span 10	5 ϕ 0.012	5 ϕ 0.012	5 ϕ 0.012	5 ϕ 0.012
			Design span 11	5 ϕ 0.012	8 ϕ 0.018	7 ϕ 0.016	5 ϕ 0.012
			Design span 12	7 ϕ 0.016	5 ϕ 0.012	9 ϕ 0.012	5 ϕ 0.012
			Design span 13	5 ϕ 0.012	5 ϕ 0.012	5 ϕ 0.012	5 ϕ 0.012
			Design span 14	6 ϕ 0.012	5 ϕ 0.012	6 ϕ 0.012	5 ϕ 0.012
			Design span 15	5 ϕ 0.012	6 ϕ 0.012	6 ϕ 0.012	5 ϕ 0.012
			Design span 16	5 ϕ 0.012	5 ϕ 0.012	5 ϕ 0.012	5 ϕ 0.012

4.2. Design from PLPost results (contour design)

- Similarly as Strip design the user should save results in PLPost then load it again in PLDesign.

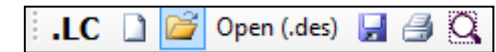
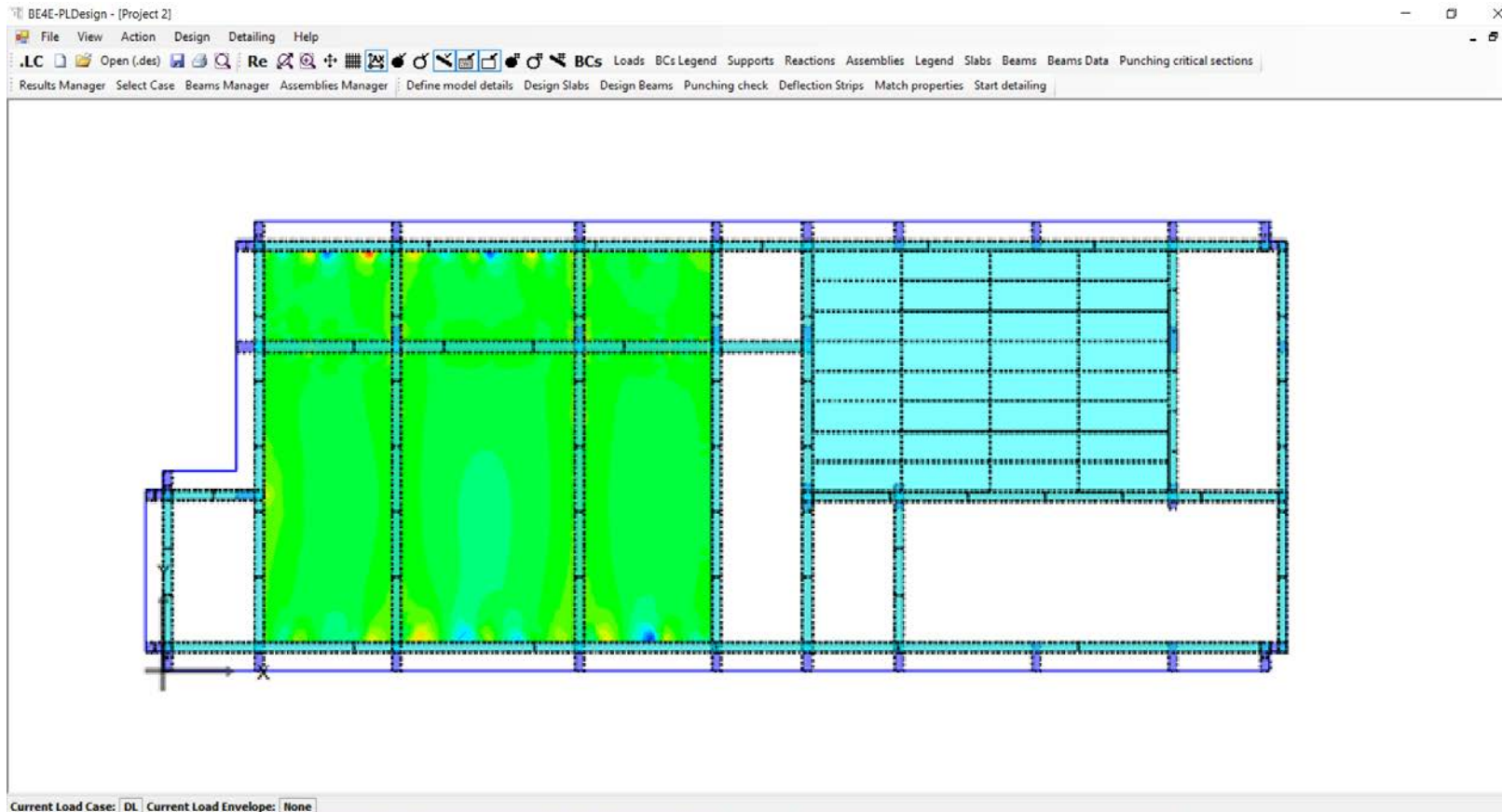


Contours

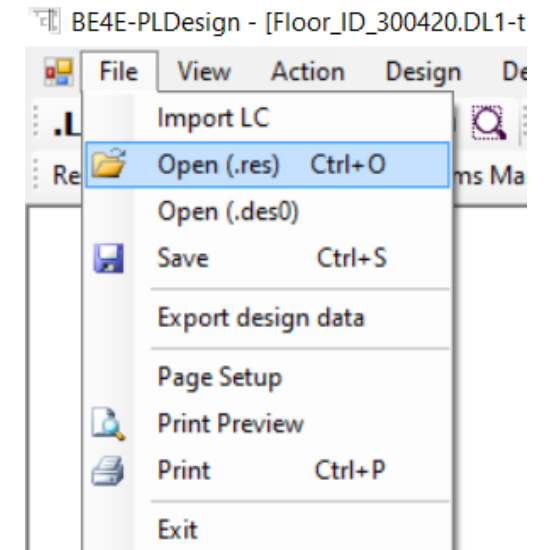
Main Contour Contour 1	X	ID: Contour 1	Enabled: <input checked="" type="checkbox"/>	Export
		N: 8	Spacing: 0.5	
		Min Is Userdefined: <input type="checkbox"/>	min: 0	
		Max Is Userdefined: <input type="checkbox"/>	max: 0	
		Status: ToBeSolved	Current Variable: Mxx	▼
		ltheta: 0		

4.2. Design from PLPost results (contour design)

- Load the saved (.res) file in PLDesign.



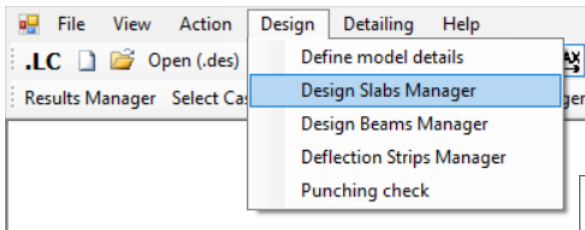
OR



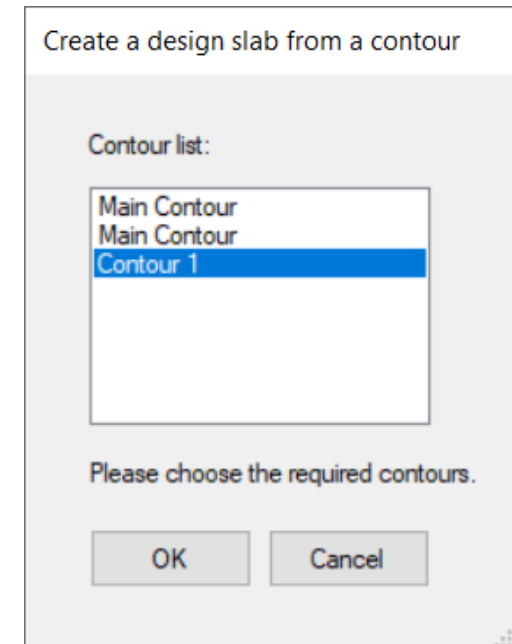
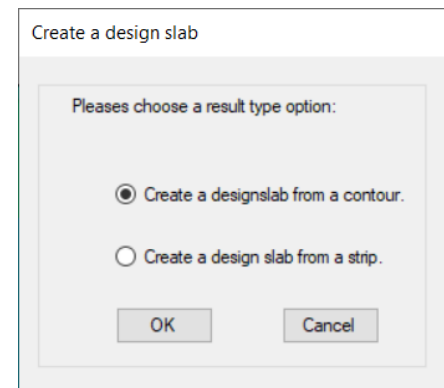
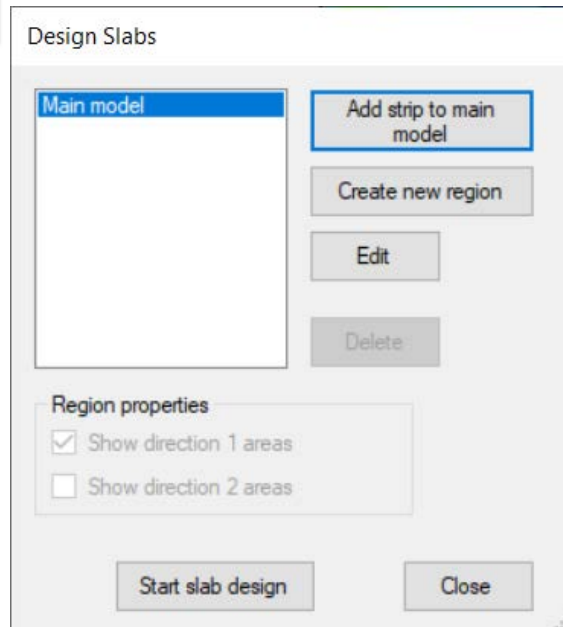
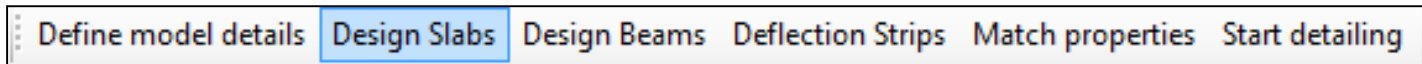
4.2. Design from PLPost results (contour design)

- Open Design Slabs Manager, then press on add strip to main model tab.
- Choose a design slab from a contour which will open a contour list to determine the contour area needed to be designed.

BE4E-PLDesign - [Project 2]

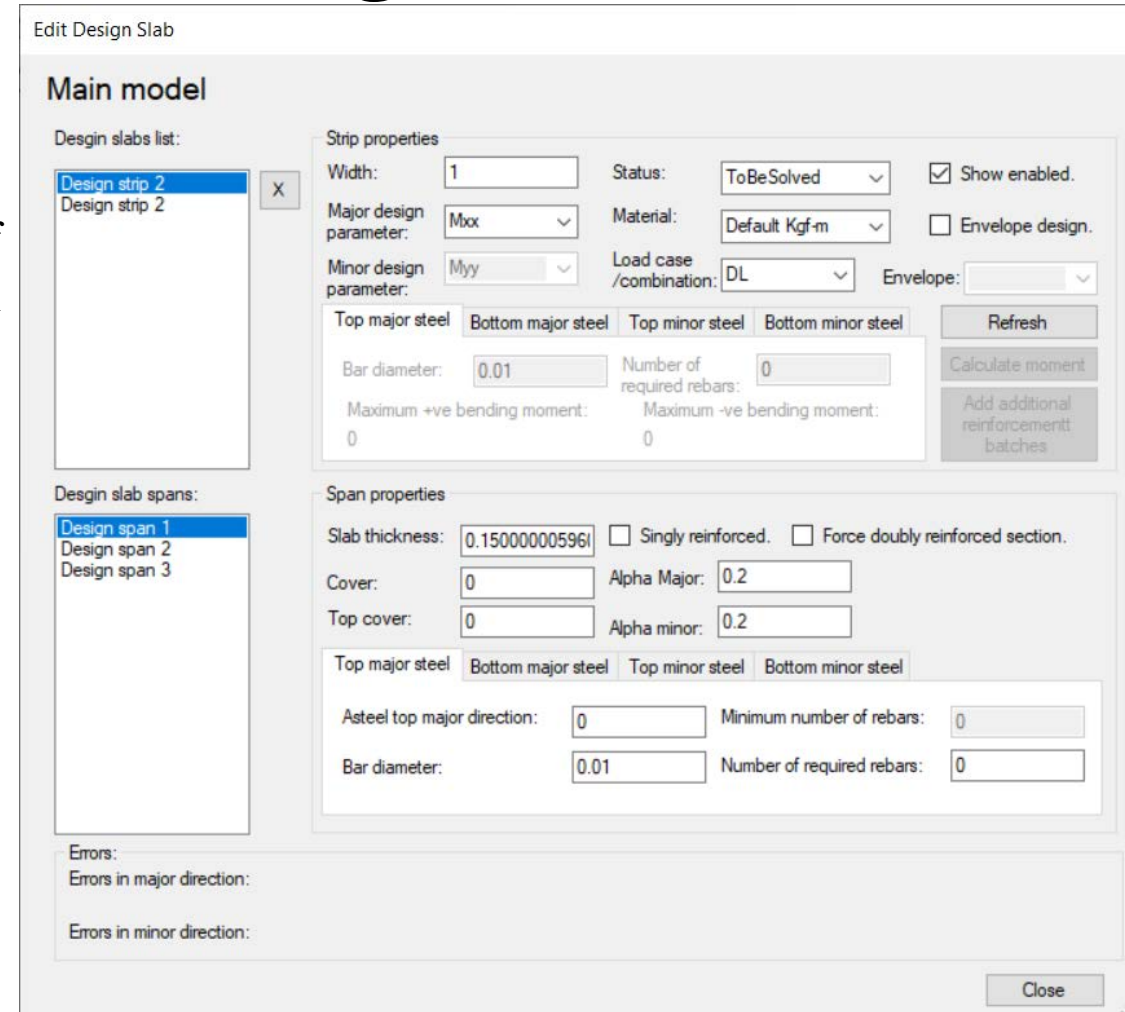
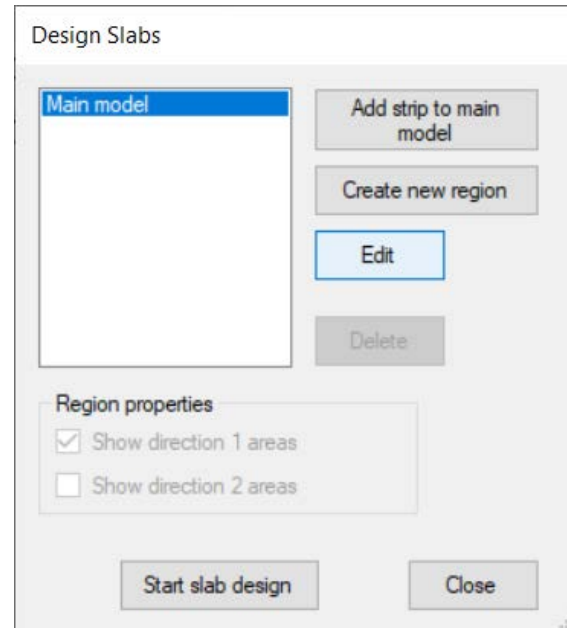


OR



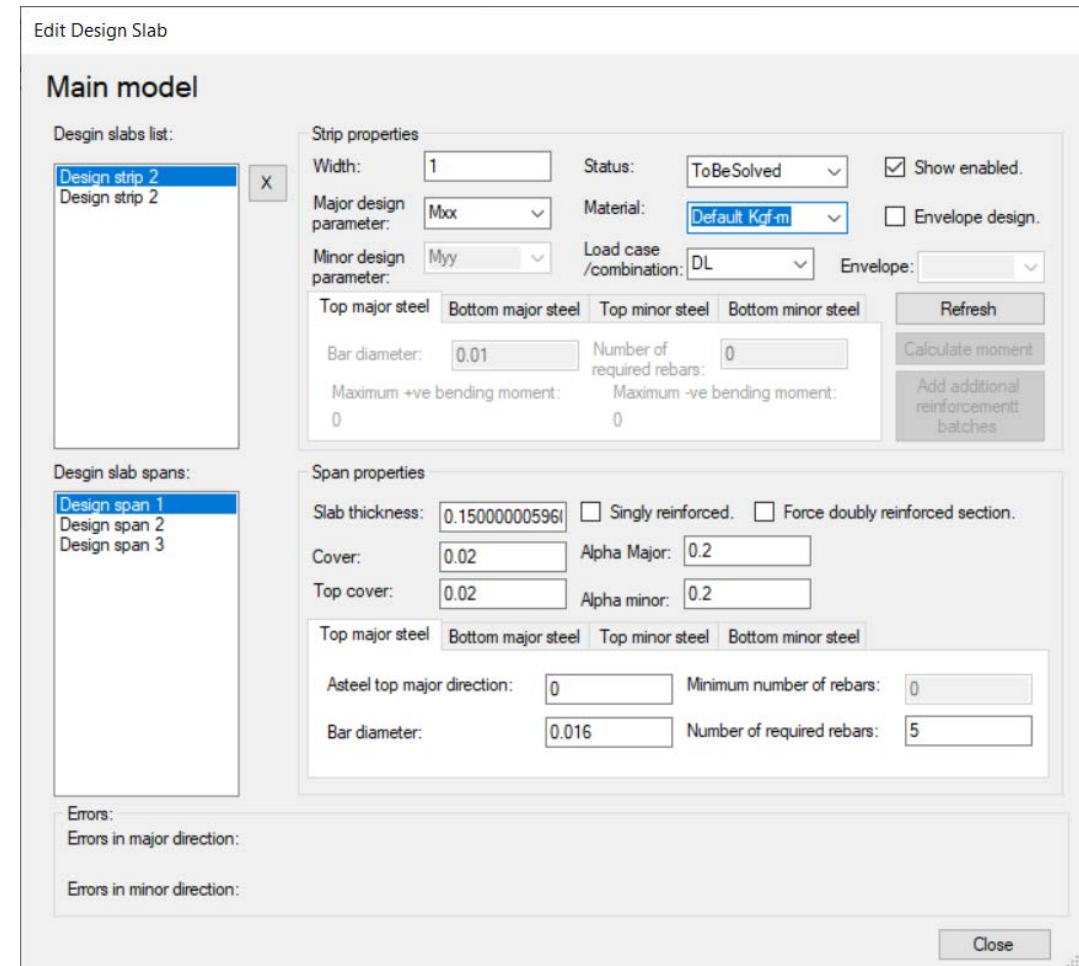
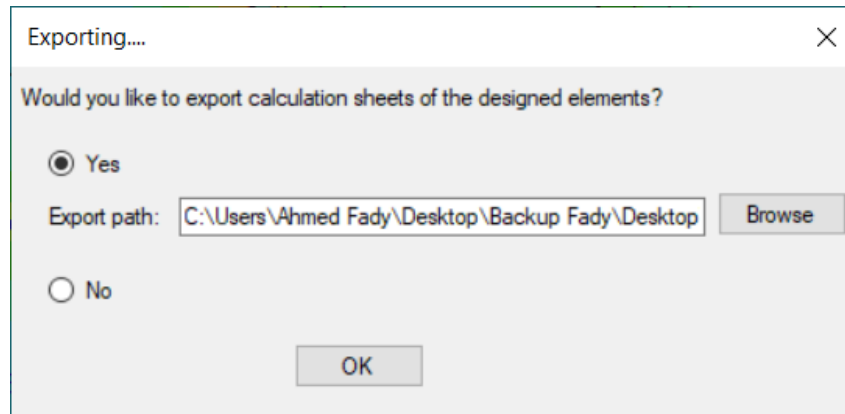
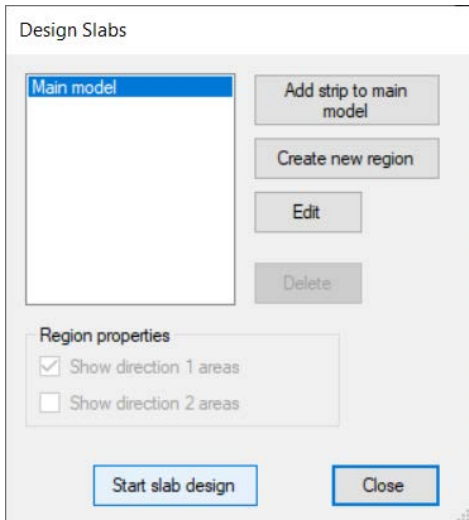
4.2. Design from PLPost results (contour design)

- Press on Edit to open Design Slab Manager.
- Similarly as strip design, the user should adjust the design slab list and design span list.
- But in contour design we note that there are 2 design strips one of them is vertical (strip 1) and the other is horizontal (strip 2) each strip should adjust it's properties.



4.2. Design from PLPost results (contour design)

- After adjusting the main model, it is time for solving the slab by pressing start slab design.
- Export the design calculation sheet.



4.2. Design from PLPost results (contour design)

- Check that all spans are safe and modify the number of bars and bar diameter if needed.
- The PLDesign is automatically export the calculation sheet for slab every section has two files one for moment in X-direction and the other for Y-direction.

Dimensions & Moment			Materials		
Moment (M)	8E+07	N.mm	Steel yield Strength (fy)	353.04	N/mm ²
Thickness of section (t)	200	mm	Concrete Cube Strength (fcu)	24.517	N/mm ²
Concrete clear cover @	10	mm	Steel Young's Modulus (E)	205340	N/mm ²
Depth of Section (d)	130	mm	Concrete Strain (ε)	0.003	
Compression st Depth (dm)	10	mm	Partial Factors	γc	1.5
				γs	1.15

Page 1

<p>Design</p> $a = \frac{(0.67 \cdot f_{cu} \cdot b \cdot d) - \sqrt{[(0.67 \cdot f_{cu} \cdot b \cdot d) - 4 \cdot (0.67 \cdot f_{cu} \cdot b \cdot d) \cdot \gamma_c \cdot M]}{2 \cdot (0.67 \cdot f_{cu} \cdot b)}$ <p>a = 45.548 mm</p> $C_{max} = \frac{2}{3} \cdot \frac{E_{cmax}}{E_{cmax} + \frac{f_y}{\gamma_s} / E_{steel}}$ <p>C_{max} = 0.4454 mm</p> <p>Check C > C_{max}</p> $c = \frac{a}{0.8/d}$ <p>C = 0.2997 mm</p> <p>Check C > dm_{max}</p> $Area\ steel = \frac{0.67 \cdot f_{cu} \cdot b \cdot a \cdot \gamma_s}{\gamma_c \cdot f_y}$ <p>Area steel = 649.3 mm² #REF!</p> $Max\ Moment = \frac{0.67 \cdot f_{cu} \cdot 0.8 \cdot b \cdot d \cdot 460 \cdot d - \frac{0.4 \cdot d \cdot 460}{690 + f_y}}{(690 + f_y) \cdot \gamma_c}$ <p>Max Moment = 114371136.4 N.mm</p> $Area\ steel\ compression = \frac{(M - M_{umax}) \cdot \gamma_s}{f_y \cdot (d - dm)}$ <p>Area steel compression = 0 mm²</p> <p>Check Area steel Maximum</p> $Area\ steel\ max = (\mu \cdot f_{cu}) \cdot (b \cdot d) + A_{compression}$ <p>Area steel max = 3260.711176 mm²</p>	<p>Check C > dm_{max}</p> <p>d'/d = 0.15</p> <p>dm_{max} = 28.5</p> <p>Check dm_{max} > dm</p>
---	---

Edit Design Slab

Main model

Design slabs list:

- Design strip 2
- Design strip 2

Strip properties

Width: Status: Show enabled.

Major design parameter: Material: Envelope design.

Minor design parameter: Load case /combination: Envelope:

Top major steel Bottom major steel Top minor steel Bottom minor steel

Bar diameter: Number of required rebars:

Maximum +ve bending moment: Maximum -ve bending moment:

Design slab spans:

- Design span 1
- Design span 2
- Design span 3

Span properties

Slab thickness: Singly reinforced. Force doubly reinforced section.

Cover: Alpha Major:

Top cover: Alpha minor:

Top major steel Bottom major steel Top minor steel Bottom minor steel

Asteel top major direction: Minimum number of rebars:

Bar diameter: Number of required rebars:

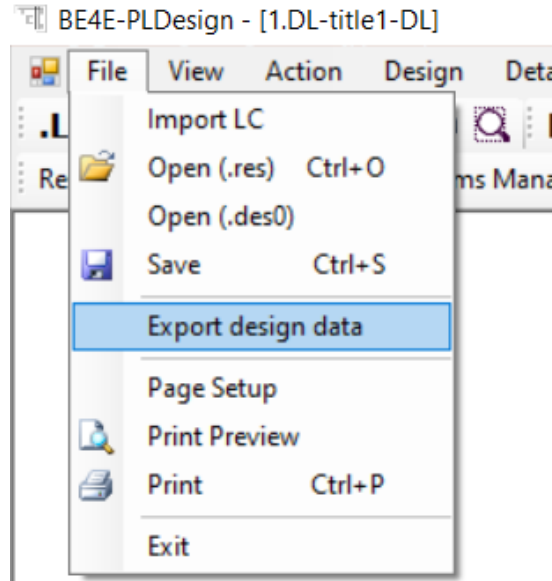
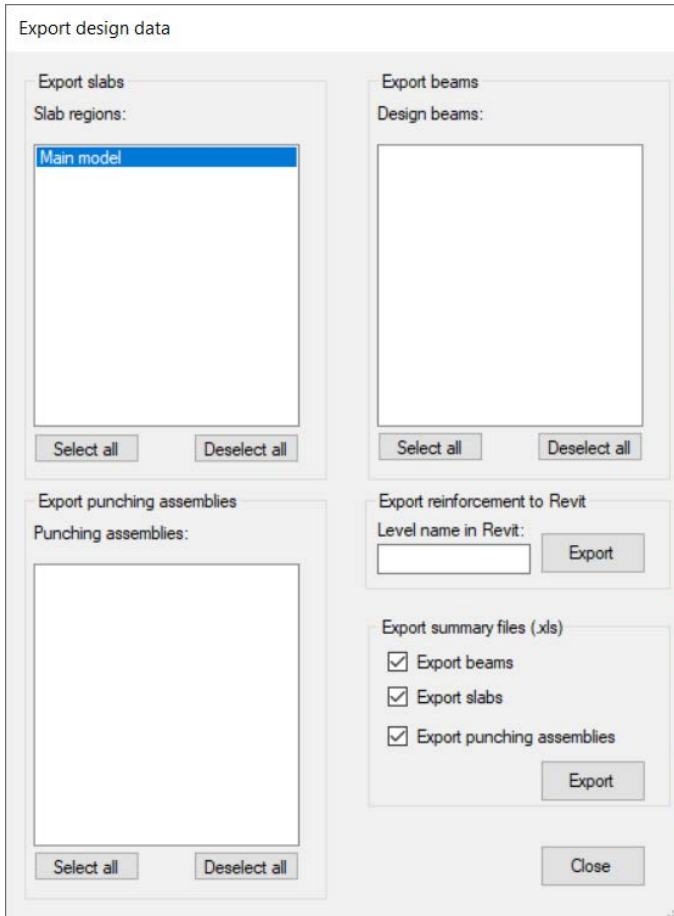
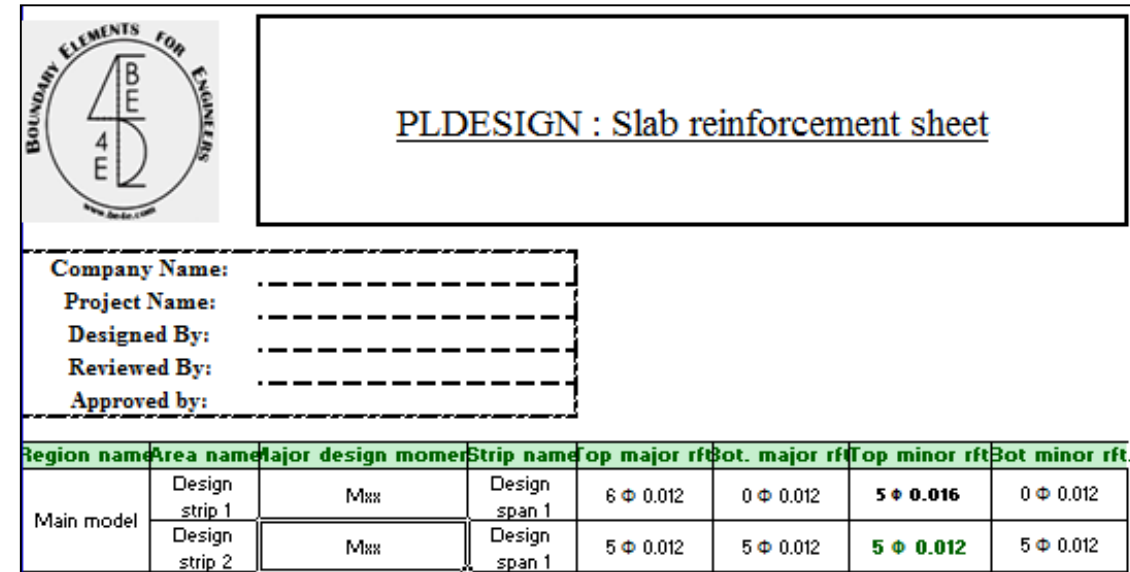
Errors:

Errors in major direction: Please force doubly reinforced section and define alpha or use a singly reinforced section.

Errors in minor direction: Please force doubly reinforced section and define alpha or use a singly reinforced section.

4.2. Design from PLPost results (contour design)

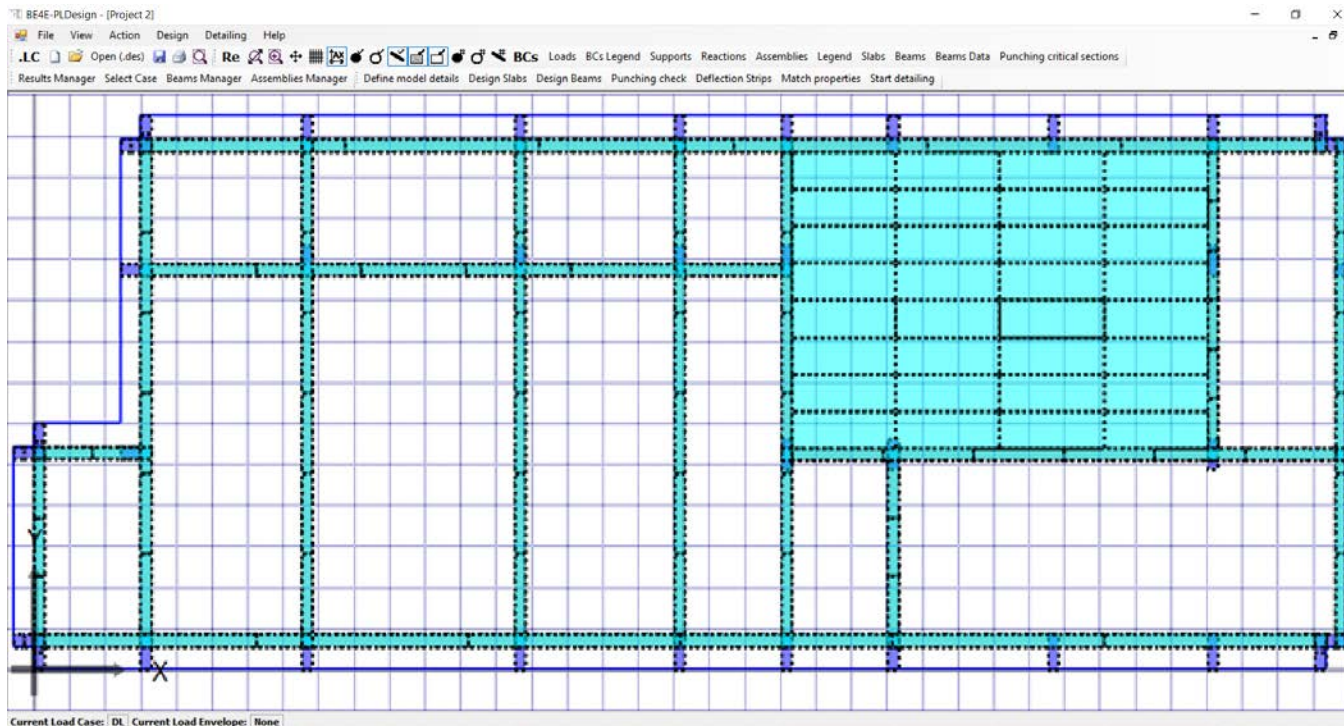
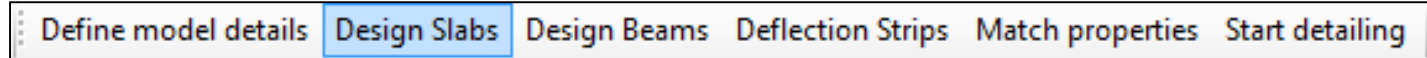
- Instead of every strip contains two excel files, the user can export a summary for slab reinforcement.

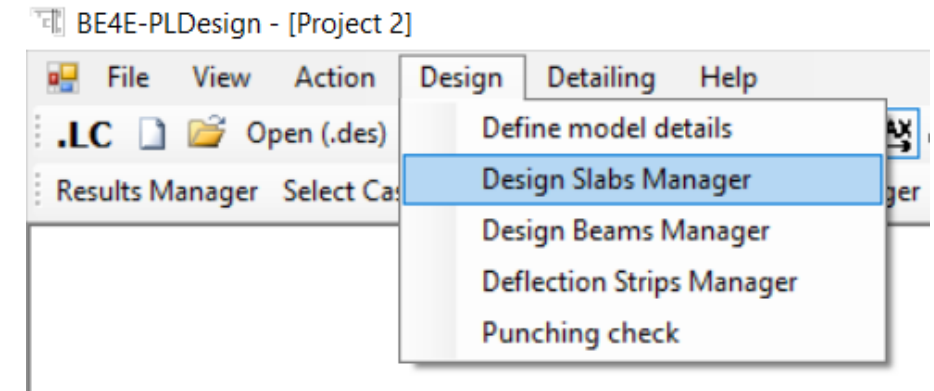
Region name	Area name	Major design momer	Strip name	op major rft	Bot. major rft	Top minor rft	Bot minor rft.
Main model	Design strip 1	Mxxx	Design span 1	6 Φ 0.012	0 Φ 0.012	5 Φ 0.016	0 Φ 0.012
	Design strip 2	Mxxx	Design span 1	5 Φ 0.012	5 Φ 0.012	5 Φ 0.012	5 Φ 0.012

4.3. Design from PLDesign directly (strip based region)

- The user has no need to use PLPost, he could use PLDesign directly.
- Design based region's idea is dividing the selected slab into number of horizontal and vertical strips, these numbers are selected by the user and could be changed according to the dimension of slab, then the PLDesign calculate automatically the straining action for the selected part and design it.

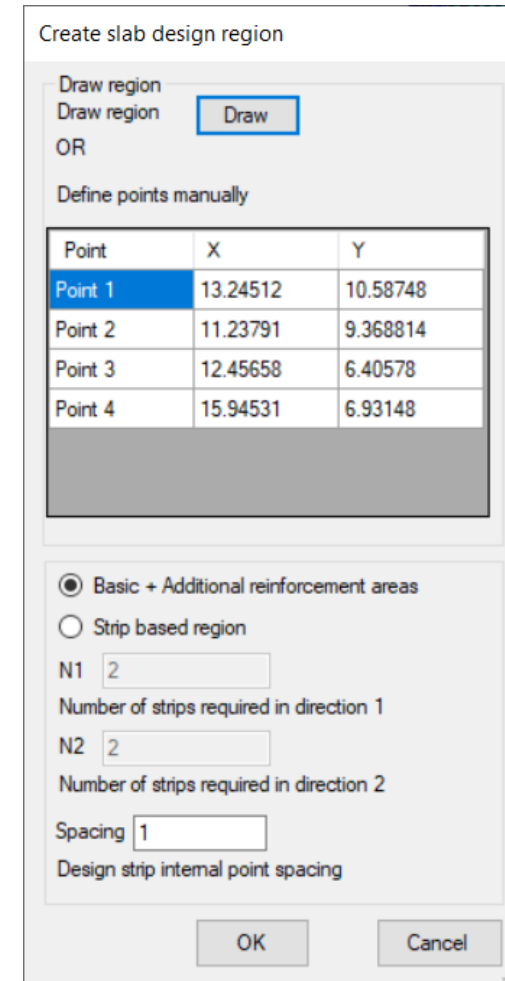
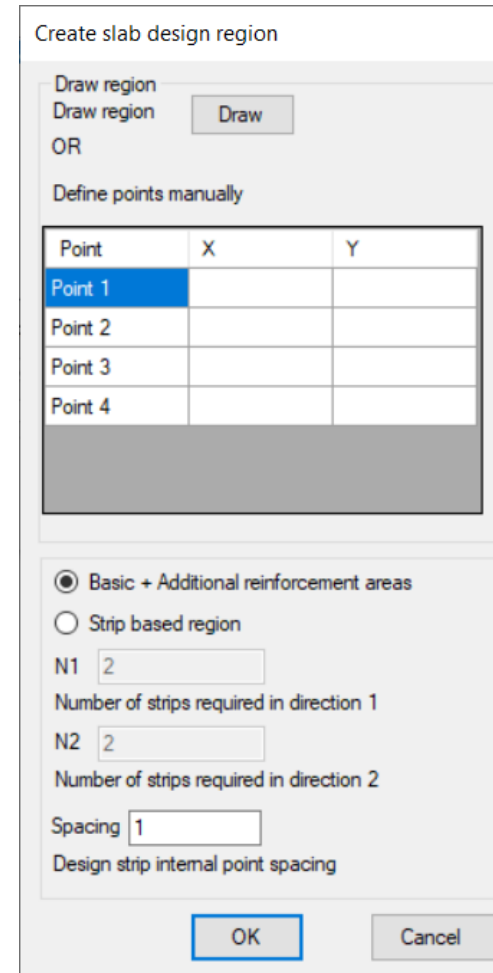
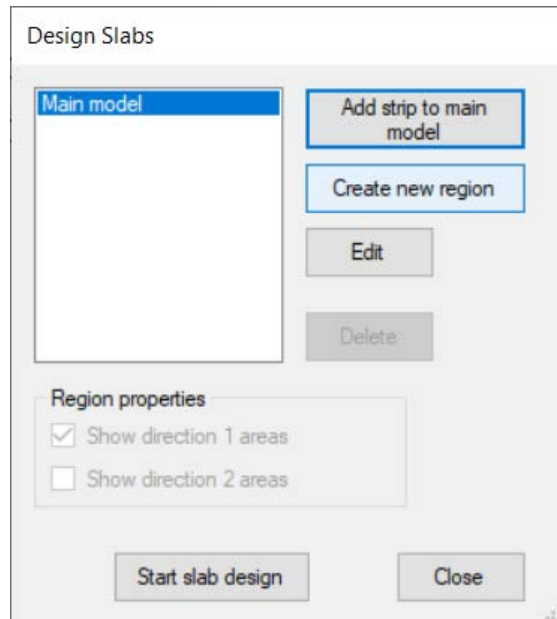


OR



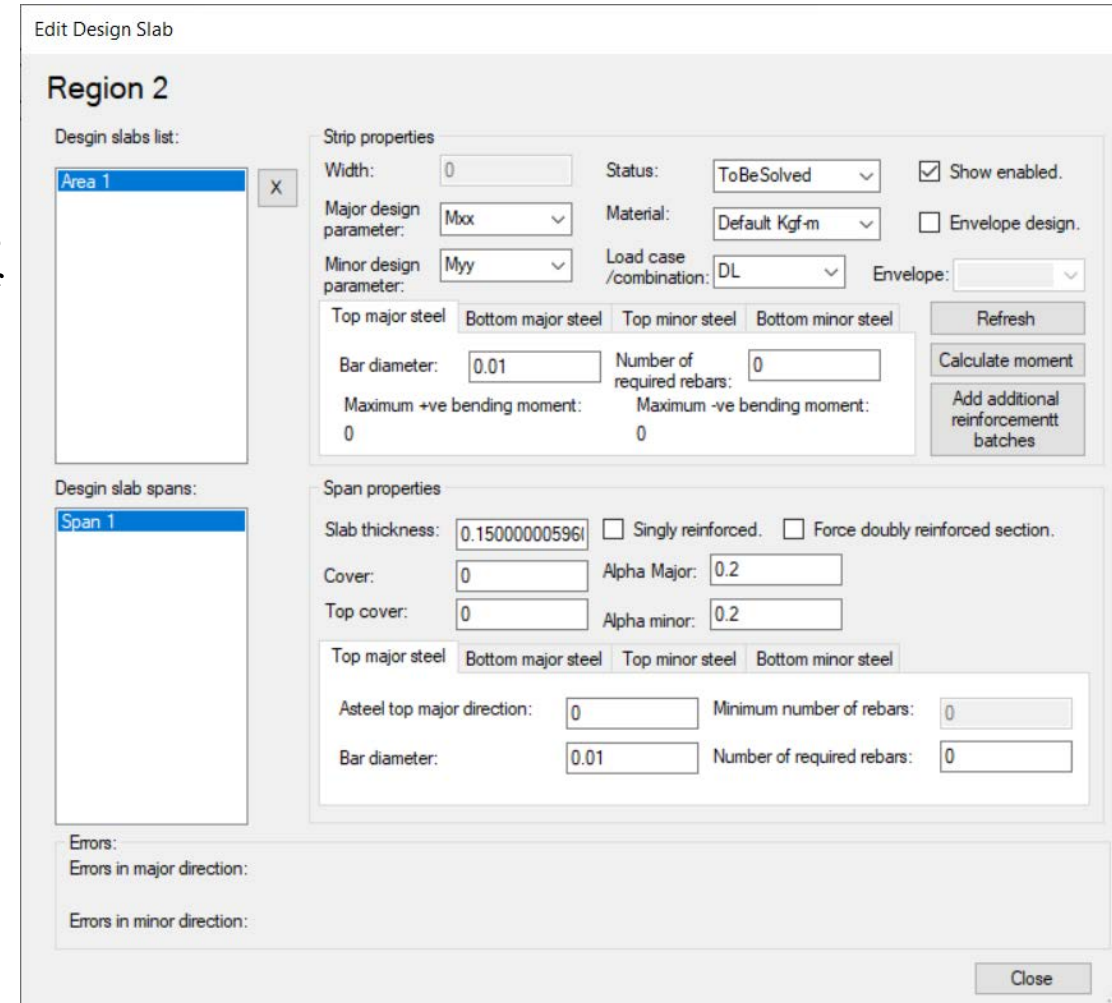
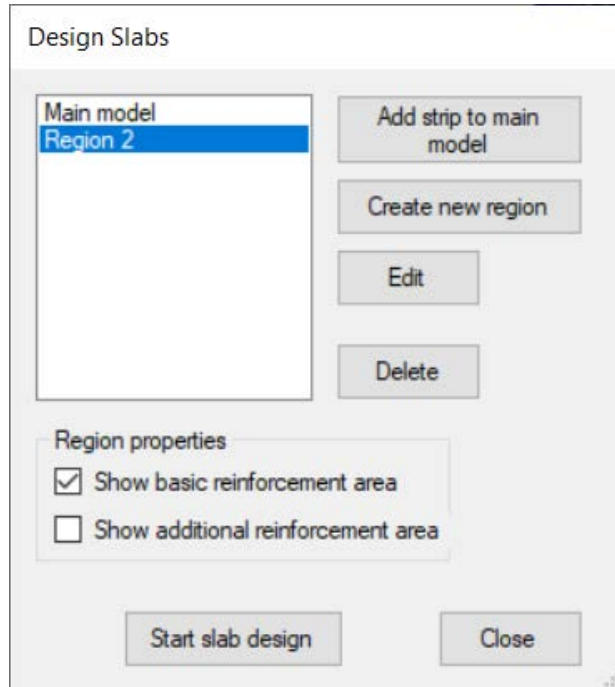
4.3. Design from PLDesign directly (strip based region)

- The two previous methods, the user use add strip to main model tab now he should use create new region as far there is no previous analysis.
- The user should draw the area needed to be designed by Draw tab, then choose Strip based region, and select the number of horizontal /vertical strips and select the spacing.



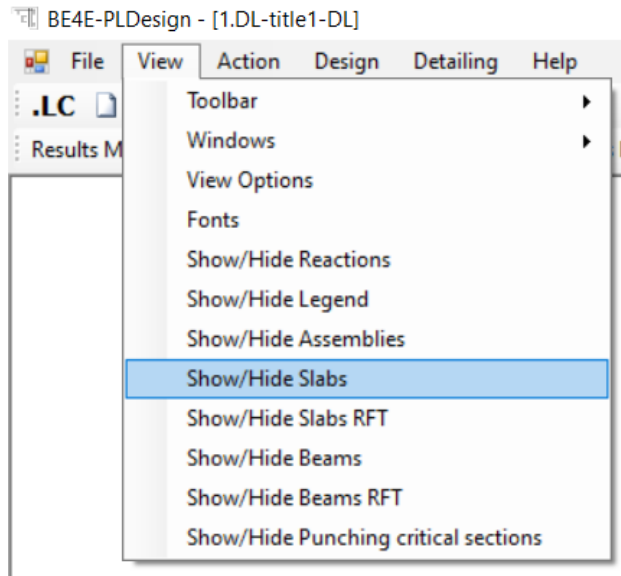
4.3. Design from PLDesign directly (strip based region)

- The PLDesign starts the analysis similarly as PLPost.
- Press on Design slab manager to see that a new region has been created.
- Go to Edit slab design to see that the slab is divided into 12 areas (8 horizontal and 4 vertical) each area divided into a number of spans.

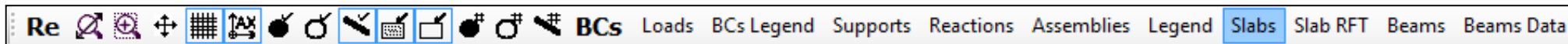
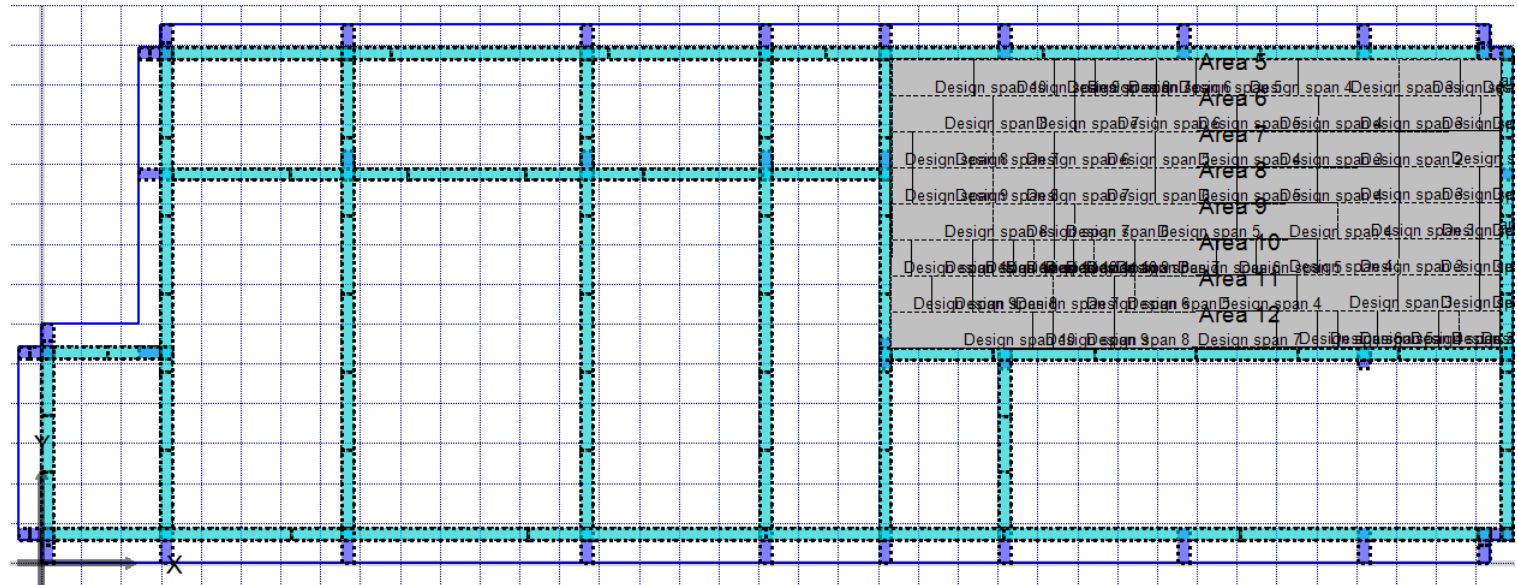


4.3. Design from PLDesign directly (strip based region)

- The horizontal areas should be designed on moment in x-direction and the vertical areas on moment in y-direction.
- The user will recognize the area and the spans in each area.



OR



4.3. Design from PLDesign directly (strip based region)

- Edit one Span in one area then match the properties for all spans.
- The user match properties twice one for areas in y-direction and the other of the spans in x-directions, then he has to check the sections before starting slab design.

Define model details Design Slabs Design Beams Deflection Strips **Match properties** Start detailing

Edit Design Slab

Region 2

Design slabs list:

- Area 1
- Area 2
- Area 3
- Area 4
- Area 5
- Area 6
- Area 7
- Area 8
- Area 9
- Area 10
- Area 11
- Area 12

Strip properties

Width: 0 Status: ToBeSolved Show enabled.

Major design parameter: Myy Material: Default Tonf Envelope design.

Minor design parameter: Myy Load case /combination: ultimate Envelope:

Top major steel Bottom major steel Top minor steel Bottom minor

Bar diameter: 0.01 Number of required rebars: 0

Maximum +ve bending moment: 0 Maximum -ve bending moment: 0

Calculate Add additional reinforcementt batches

Span properties

Slab thickness: 0000298023224 Singly reinforced. Force doubly reinforced section.

Cover: 0.01 Alpha Major: 0.2

Top cover: 0.01 Alpha minor: 0.2

Top major steel Bottom major steel Top minor steel Bottom minor steel

Asteel top major direction: 0 Minimum number of rebars: 0

Bar diameter: 0.012 Number of required rebars: 5

Errors:

Errors in major direction:

Errors in minor direction:

Close

Edit Design Slab

Region 2

Design slabs list:

- Area 1
- Area 2
- Area 3
- Area 4
- Area 5
- Area 6
- Area 7
- Area 8
- Area 9
- Area 10
- Area 11
- Area 12

Strip properties

Width: 0 Status: ToBeSolved Show enabled.

Major design parameter: Mxx Material: Default Tonf Envelope design.

Minor design parameter: Myy Load case /combination: ultimate Envelope:

Top major steel Bottom major steel Top minor steel Bottom minor

Bar diameter: 0.01 Number of required rebars: 0

Maximum +ve bending moment: 0 Maximum -ve bending moment: 0

Calculate Add additional reinforcementt batches

Design slab spans:

- Design span 1
- Design span 2
- Design span 3
- Design span 4
- Design span 5
- Design span 6
- Design span 7
- Design span 8
- Design span 9
- Design span 10

Span properties

Slab thickness: 0000298023224 Singly reinforced. Force doubly reinforced section.

Cover: 0.01 Alpha Major: 0.2

Top cover: 0.01 Alpha minor: 0.2

Top major steel Bottom major steel Top minor steel Bottom minor steel

Asteel top major direction: 0 Minimum number of rebars: 0

Bar diameter: 0.012 Number of required rebars: 5

Errors:

Errors in major direction:

Errors in minor direction:

Close

Match properties

Slab spans Beams Beam sections Punching asms.

Source region : Region 2 Destination region: Region 2

Source area : Area 1 Destination area: Area 3

Source span: Design span 1 Design span 2 Destination span: Design span 1 Design span 2 Design span 3 Design span 4

Top major steel Bar diameters Bar amounts

Top minor steel Bar diameters Number of bars

Bottom major steel Bar diameters Number of bars

Top major steel Bar diameters Number of bars

Dimensions Slab thickness Bottom cover Top cover

Section data Is Singly reinforced Force doubly reinforced section. Alpha values

Match slabs

Close

4.3. Design from PLDesign directly (strip based region)

- Before starting slab design, the user should check mark on show direction 2 areas.

Design Slabs

Main model
Region 2

Add strip to main model

Create new region

Edit

Delete

Region properties

Show basic reinforcement area

Show additional reinforcement area

Start slab design

Close

Dimensions & Moment

Moment (M)	836760	N.mm
Thickness of section (t)	200	mm
Concrete clear cover ϕ	10	mm
Depth of Section (d)	190	mm

Materials

Steel yield Strength (fy)	353.039	N/mm ²
Concrete Cube Strength (fcu)	24.5166	N/mm ²
Steel Young's Modulus (E)	205940	N/mm ²
Concrete Strain (ϵ)	0.003	
Partial Factors	γ_c : 1.5 γ_s : 1.15	

Design

$$a = \frac{(0.67 \cdot f_{cu} \cdot b \cdot d) - \sqrt{[(0.67 \cdot f_{cu} \cdot b \cdot d) - 4 \cdot (0.67 \cdot f_{cu} \cdot b \cdot d) \cdot (1 - M/2)]}}{2 \cdot (0.67 \cdot f_{cu} \cdot b)}$$

a = 19.0000003 mm

$$c_{max} = 2/3 \cdot \frac{E_{cmax}}{E_{cmax} + \frac{f_y}{\gamma_s} / E_{steel}}$$

c_{max} = 0.445366528 mm

Check C < C_{max}

$$c = \frac{a}{0.8/d}$$

c = 0.125 mm

$$Area\ steel = \frac{0.67 \cdot f_{cu} \cdot b \cdot a \cdot \gamma_s}{\gamma_c \cdot f_y}$$

A_s = 677.7546403 mm²

Check Area steel Maximum

$$Area\ steel\ max = (\mu \cdot f_{cu}) \cdot (b \cdot d)$$

Area steel max = 3260.7 mm² A_{smax} > A_s

Check Area Steel Minimum

Area 5	M _{xxx}	Design span 1	5 ϕ 0.012	6 ϕ 0.012	5 ϕ 0.012	6 ϕ 0.012
		Design span 2	5 ϕ 0.012	6 ϕ 0.012	5 ϕ 0.012	6 ϕ 0.012
		Design span 3	6 ϕ 0.012	5 ϕ 0.012	6 ϕ 0.012	5 ϕ 0.012
		Design span 4	5 ϕ 0.012	6 ϕ 0.012	5 ϕ 0.012	6 ϕ 0.012
		Design span 5	7 ϕ 0.012	5 ϕ 0.012	5 ϕ 0.012	6 ϕ 0.012
		Design span 6	5 ϕ 0.012	7 ϕ 0.012	5 ϕ 0.012	6 ϕ 0.012
		Design span 7	5 ϕ 0.012	6 ϕ 0.012	6 ϕ 0.012	5 ϕ 0.012
		Design span 8	6 ϕ 0.012	5 ϕ 0.012	6 ϕ 0.012	5 ϕ 0.012
		Design span 9	6 ϕ 0.012	5 ϕ 0.012	6 ϕ 0.012	5 ϕ 0.012
		Design span 10	6 ϕ 0.012	5 ϕ 0.012	5 ϕ 0.012	6 ϕ 0.012
Area 6	M _{xxx}	Design span 1	6 ϕ 0.012	5 ϕ 0.012	6 ϕ 0.012	5 ϕ 0.012
		Design span 2	5 ϕ 0.012	6 ϕ 0.012	5 ϕ 0.012	6 ϕ 0.012
		Design span 3	6 ϕ 0.012	5 ϕ 0.012	5 ϕ 0.012	6 ϕ 0.012
		Design span 4	5 ϕ 0.012	6 ϕ 0.012	5 ϕ 0.012	6 ϕ 0.012
		Design span 5	6 ϕ 0.012	5 ϕ 0.012	5 ϕ 0.012	6 ϕ 0.012
		Design span 6	5 ϕ 0.012	6 ϕ 0.012	6 ϕ 0.012	5 ϕ 0.012
		Design span 7	6 ϕ 0.012	5 ϕ 0.012	6 ϕ 0.012	5 ϕ 0.012
		Design span 8	5 ϕ 0.012	6 ϕ 0.012	5 ϕ 0.012	6 ϕ 0.012

Exporting...

Would you like to export calculation sheets of the designed elements?

Yes

Export path: C:\Users\Ahmed Fady\Desktop\Backup Fady\Desktop

No

OK

4.3. Design from PLDesign directly (strip based region)

- One of PLDesign advantages is the detailing where the user after finishing the design, he can see slab detailing.
- The user will have (.Dxf) file, the file contain two drawings one for horizontal areas and the other for vertical areas.

Start detailing

Slab detailing

Main model
Region 2

Layer name	Content	Color	Export
▶ Slab areas	Slab areas	Red	<input checked="" type="checkbox"/>
Major top rft.	Major top rft.	Blue	<input checked="" type="checkbox"/>
Major bot rft.	Major bot rft.	Green	<input checked="" type="checkbox"/>
Minor top rft.	Minor top rft.	Yellow	<input checked="" type="checkbox"/>
Minor bot rft.	Minor bot rft.	Cyan	<input checked="" type="checkbox"/>

Select all Deselect all

Export area steel

Beam detailing

Layer name	Content	Color	Export
▶ Beam Layout	Beam layout	Red	<input checked="" type="checkbox"/>
Beam longitudinal...	Beam longitu...	Blue	<input checked="" type="checkbox"/>
Beam cross secti...	Beam cross s...	Green	<input checked="" type="checkbox"/>

Select all Deselect all

Text height-headers: 0.15 Text height-details: 0.05 Export Close

Define model details Design Slabs Design Beams Deflection Strips Match properties **Start detailing**

OR

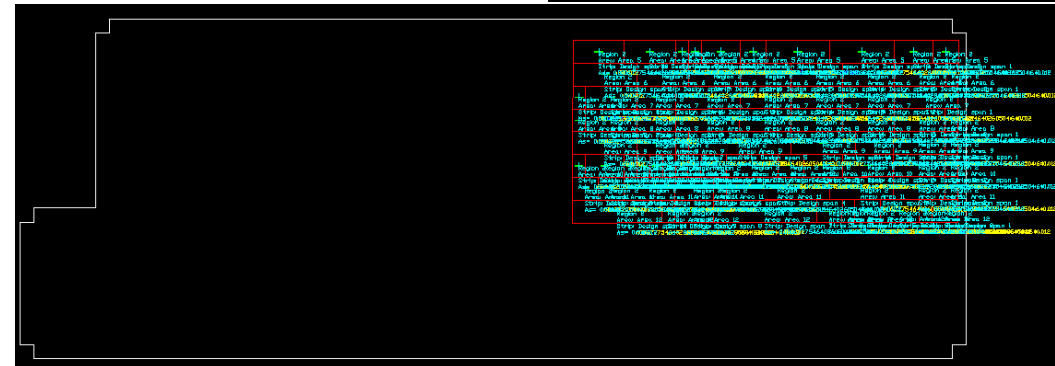
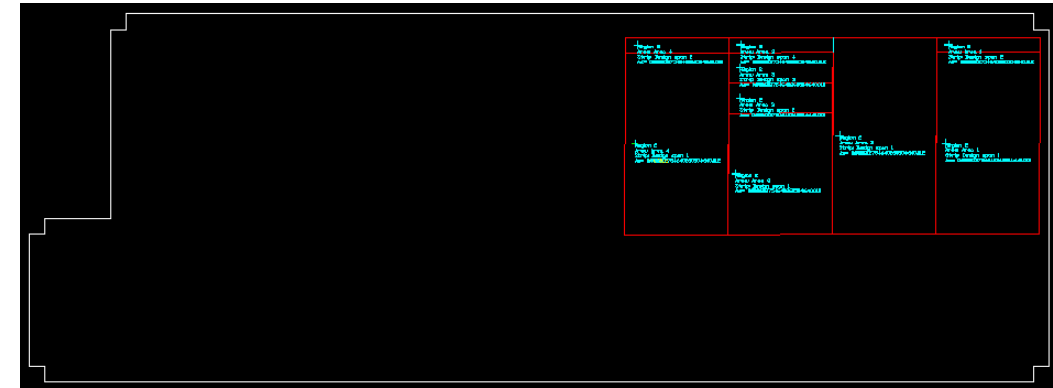
BE4E-PLDesign - [1.DL-title1-DL]

File View Action Design Detailing Help

.LC Open (.des)

Start detailing

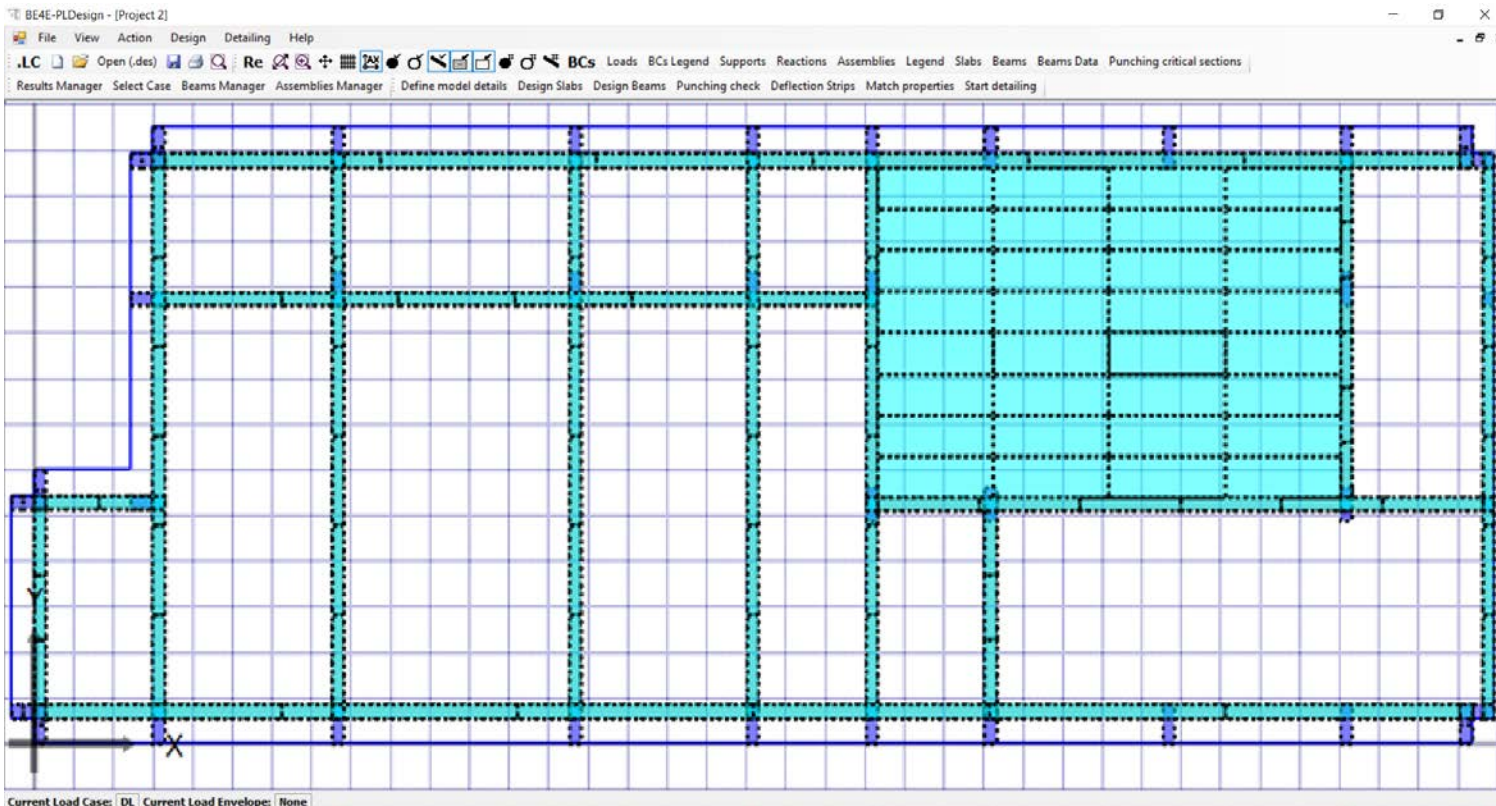
Results Manager Select Case Beams Manager Assemblies Manager



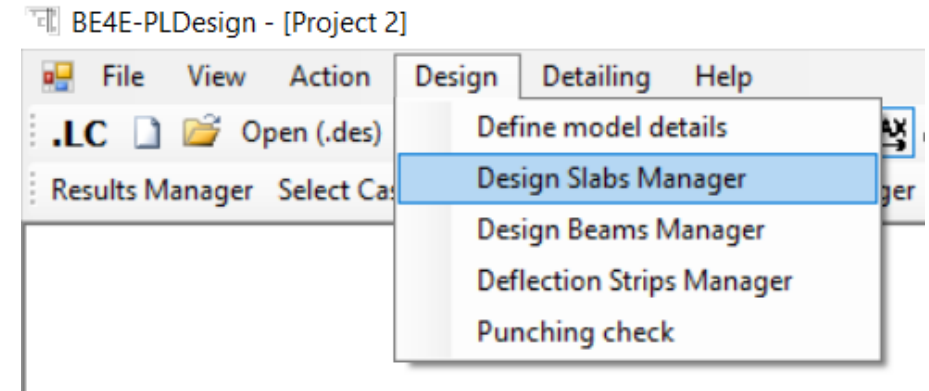
4.4 Design from PLDesign directly (basic and additional RFT)

- This method is very famous in flat slabs and can be shown very simply.

Define model details Design Slabs Design Beams Deflection Strips Match properties Start detailing

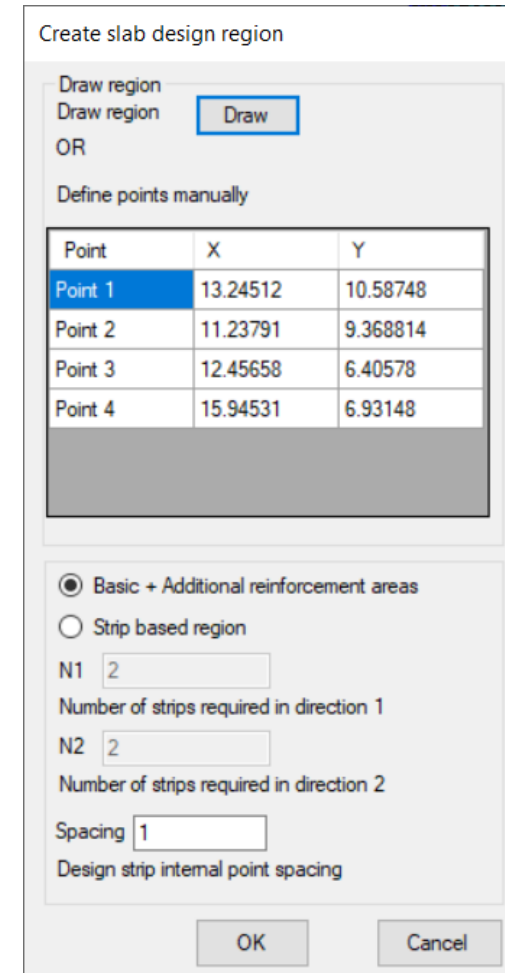
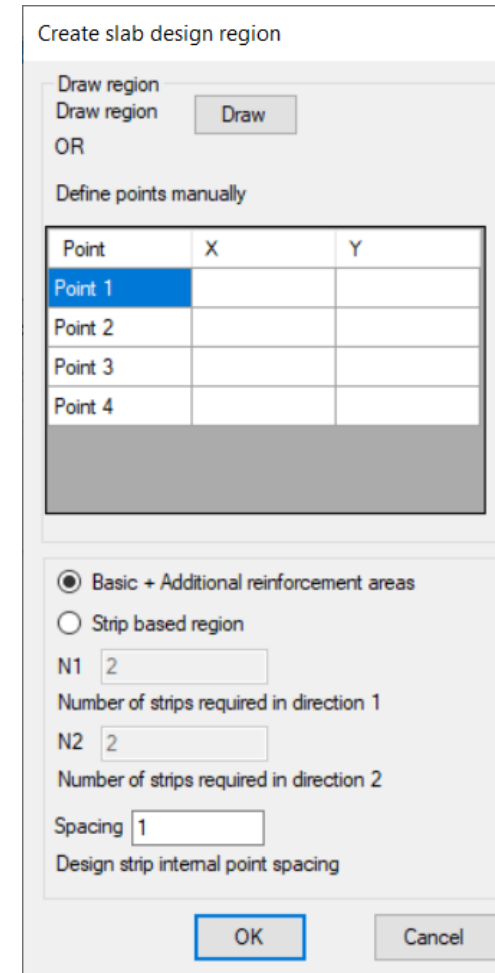
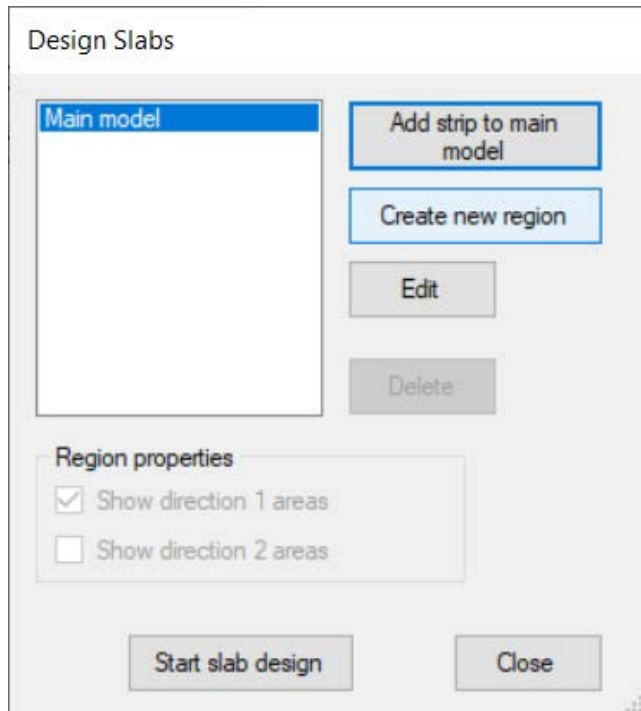


OR



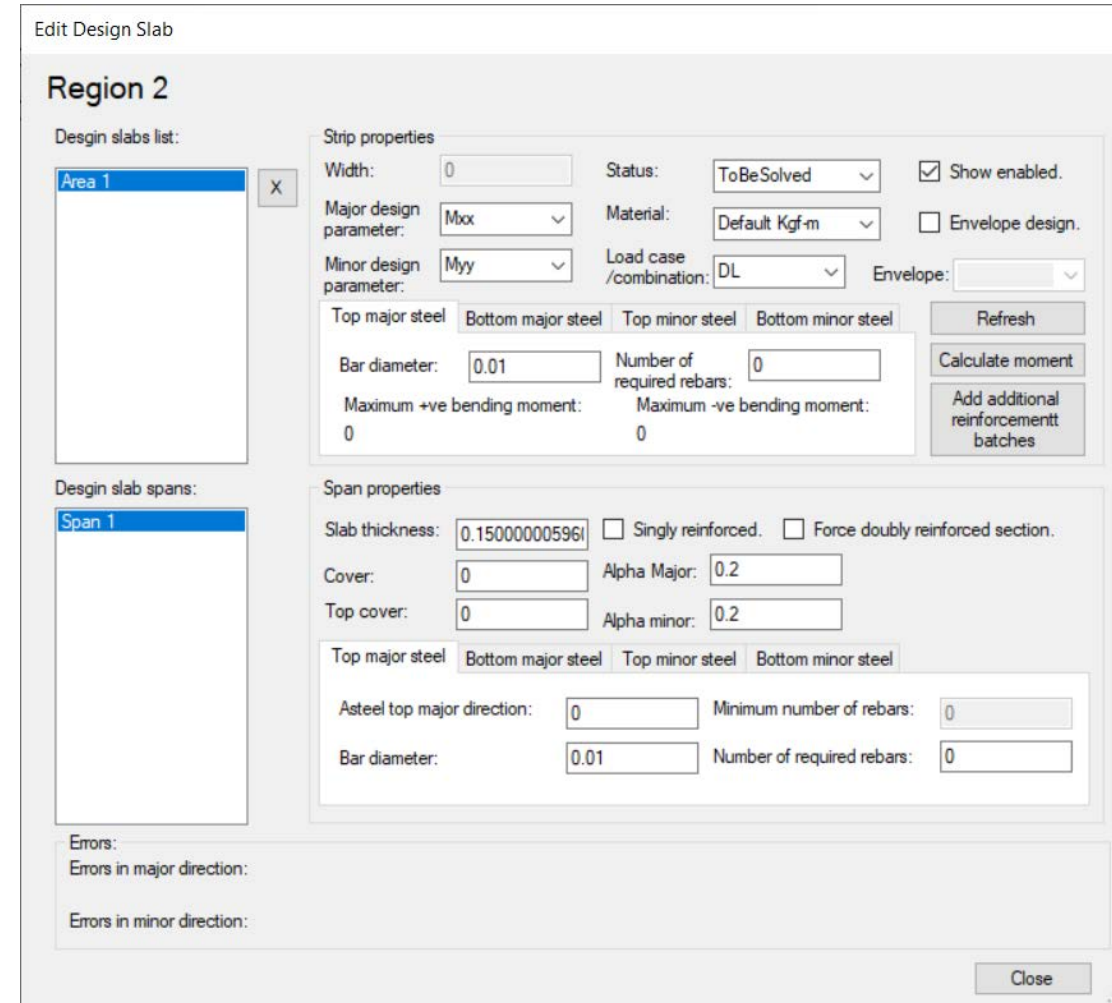
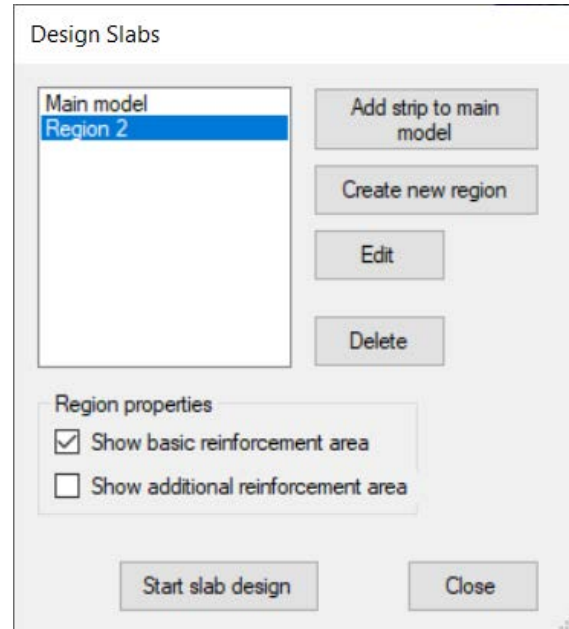
4.4 Design from PLDesign directly (basic and additional RFT)

- We are also going to use Create new region.
- The user should draw the area needed to be designed by Draw tab, then select the spacing for the analysis.



4.4 Design from PLDesign directly (basic and additional RFT)

- The PLDesign starts the analysis similarly as PLPost.
- Press on Design slab manager to see that a new region has been created.
- Go to Edit slab design to see that the slab is one area.
- The upper section of strip properties is for the basic reinforcement and the lower is for additional reinforcement.



4.4 Design from PLDesign directly (basic and additional RFT)

- The user has to insert the major/minor design parameter, Load case/combination, material units, Bar diameter and number of bars, then press calculate and refresh.
- The PLDesign calculate the maximum +ve and -ve moments.
- Click on add additional reinforcement batches to show areas for additional reinforcement.
- Press on add additional areas then draw to show parts of slab need additional reinforcement.

Create additional reinforcement areas

Areas list

Draw additional reinforcement area
Draw region
OR
Define points manually

Point	X	Y
Point 1		
Point 2		
Point 3		
Point 4		

Edit Design Slab

Region 2

Design slabs list:

Area 1

Strip properties

Width: Status: Show enabled.
 Major design parameter: Material: Envelope design.
 Minor design parameter: Load case /combination: Envelope:

Top major steel Bottom major steel Top minor steel Bottom minor steel

Bar diameter: Number of required rebars:

Maximum +ve bending moment: Reinforcement mesh too small
 Maximum -ve bending moment: Reinforcement mesh too small

Design slab spans:

Span 1

Span properties

Slab thickness: Singly reinforced. Force doubly reinforced section.
 Cover: Alpha Major:
 Top cover: Alpha minor:

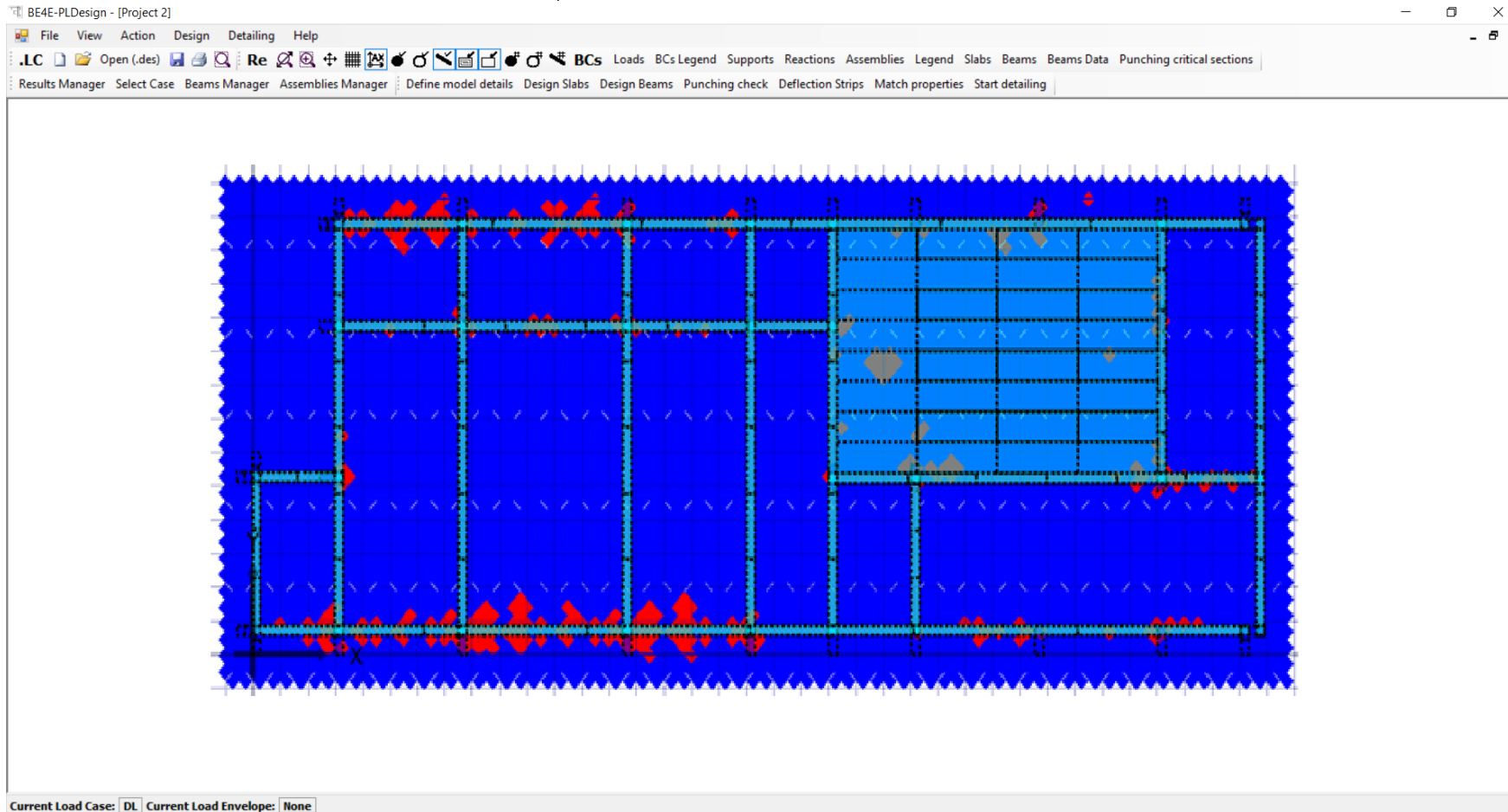
Top major steel Bottom major steel Top minor steel Bottom minor steel

Asteel top major direction: Minimum number of rebars:
 Bar diameter: Number of required rebars:

Errors:
 Errors in major direction:
 Errors in minor direction:

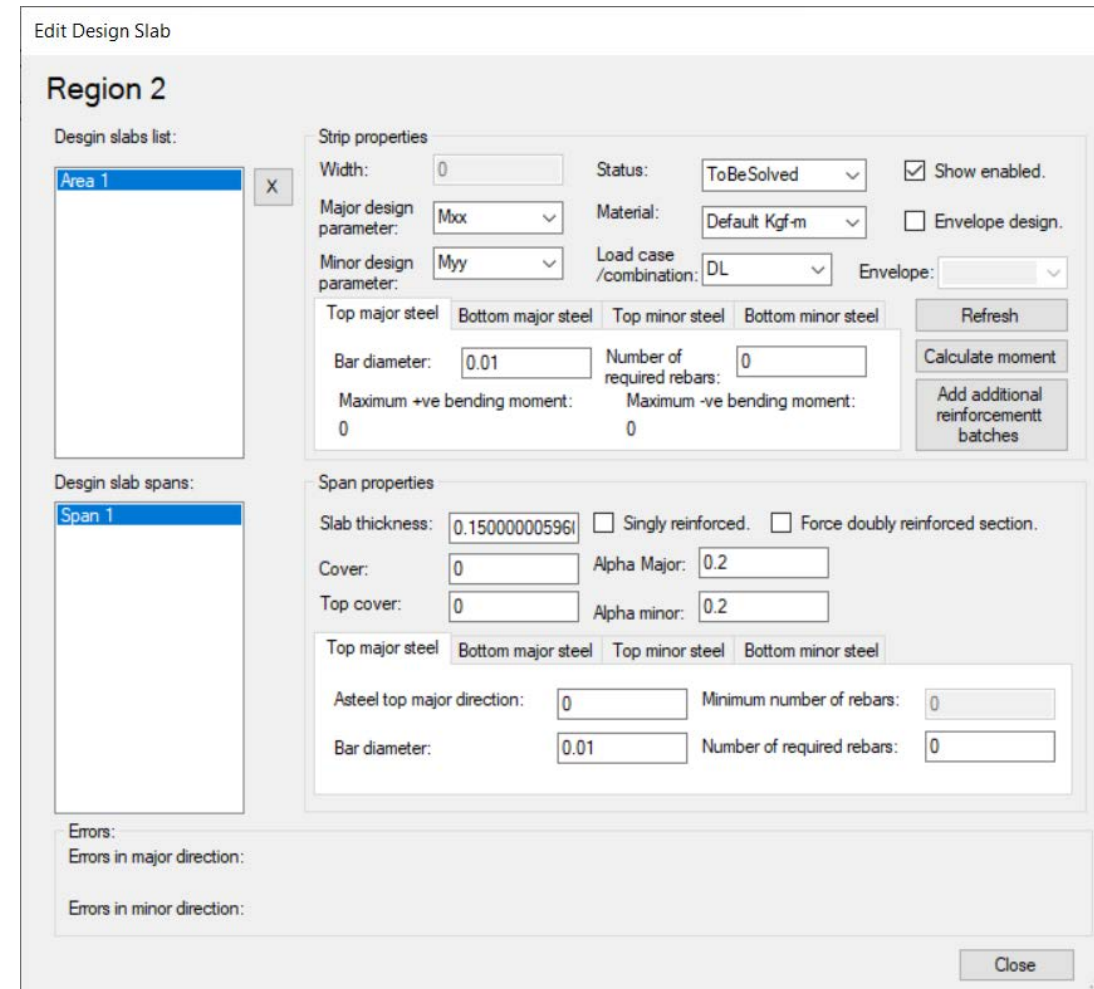
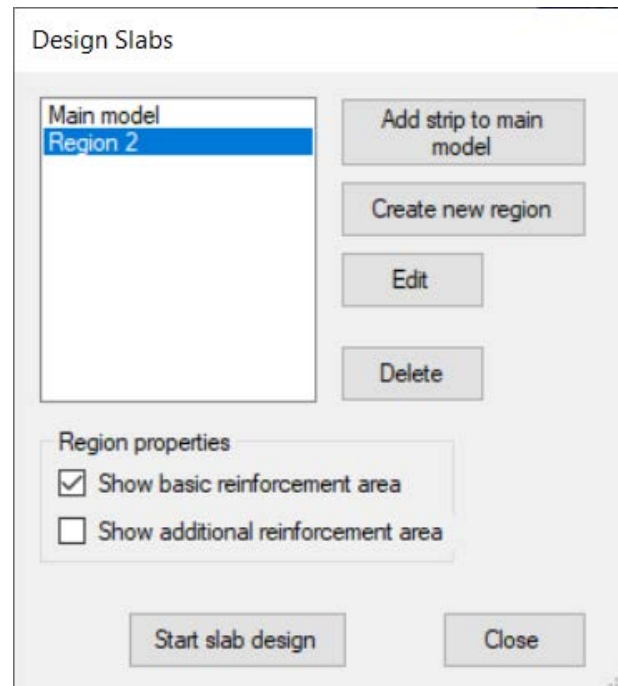
4.4 Design from PLDesign directly (basic and additional RFT)

- The blue area is safe for the basic reinforcement, but the red areas are not safe.



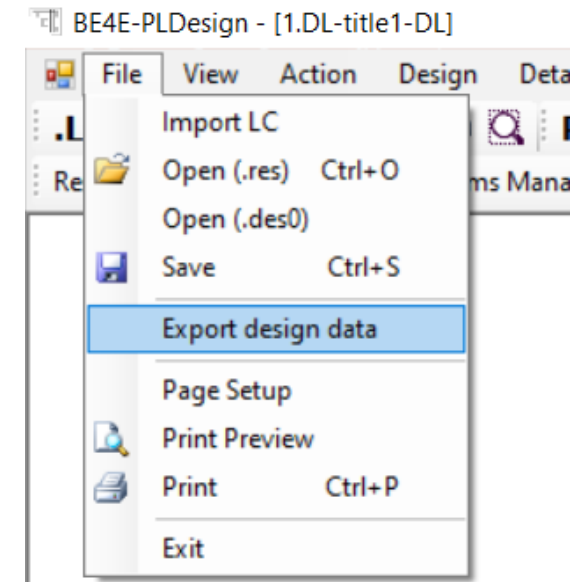
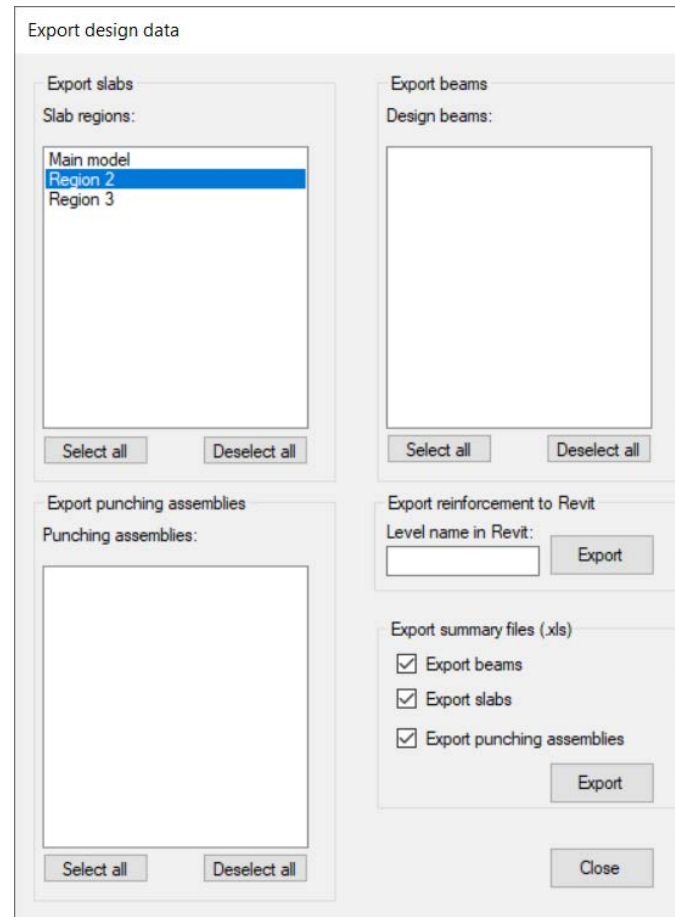
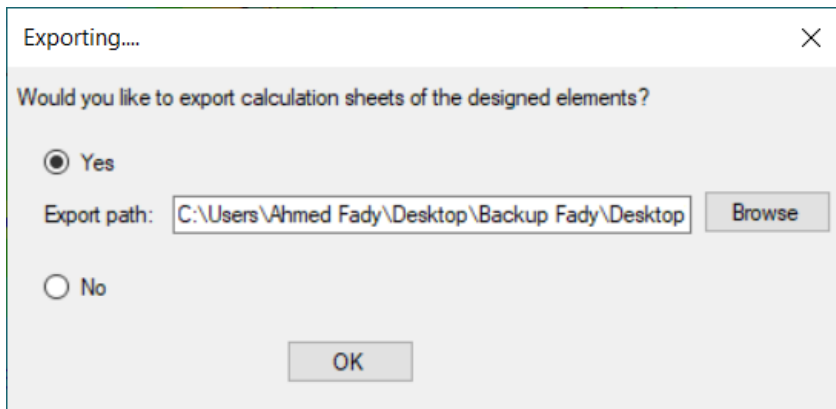
4.4 Design from PLDesign directly (basic and additional RFT)

- After drawing the additional areas, the user should select cover, bar diameter and number of required bars in span properties.
- Check the mark box of show additional reinforcement area, then Start slab design.



4.4 Design from PLDesign directly (basic and additional RFT)

- Export the calculation sheet files, then check the reinforcement and export the data sheet summary.



4.4 Design from PLDesign directly (basic and additional RFT)

- Now the user can also see slab detailing as strip based region.



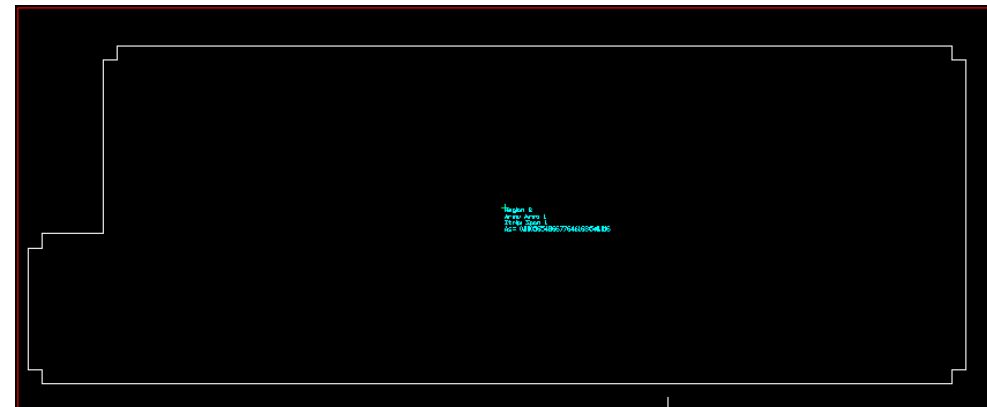
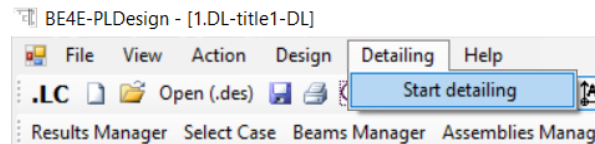
PLDESIGN : Slab reinforcement sheet

Company Name:
Project Name:
Designed By:
Reviewed By:
Approved by:

Region name	Area name	Major design moment	Strip name	Top major rft.	Bot. major rft.	Top minor rft.	Bot minor rft.
Region 2	Area 1	Mxx	Span 1	5 Φ 0.016	5 Φ 0.016	5 Φ 0.016	5 Φ 0.016

Define model details Design Slabs Design Beams Deflection Strips Match properties **Start detailing**

OR



Start detailing

Slab detailing

Main model
Region 2

Layer name	Content	Color	Export
Slab areas	Slab areas	Red	<input checked="" type="checkbox"/>
Major top rft.	Major top rft.	Blue	<input checked="" type="checkbox"/>
Major bot rft.	Major bot rft.	Green	<input checked="" type="checkbox"/>
Minor top rft.	Minor top rft.	Yellow	<input checked="" type="checkbox"/>
Minor bot rft.	Minor bot rft.	Cyan	<input checked="" type="checkbox"/>

Select all Deselect all

Export area steel

Beam detailing

Layer name	Content	Color	Export
Beam Layout	Beam layout	Red	<input checked="" type="checkbox"/>
Beam longitudinal...	Beam longitu...	Blue	<input checked="" type="checkbox"/>
Beam cross secti...	Beam cross s...	Green	<input checked="" type="checkbox"/>

Select all Deselect all

Text height-headers: 0.15 Text height-details: 0.05 Export Close

PLDesign Package

2.1. File needed to be exported before using PLDesign ✓

2.2. Starting PLDesign ✓

2.3. Load combinations & load envelopes ✓

2.4. Slab design ✓

2.4.1. Design from PLPost results (strip design) ✓

2.4.2. Design from PLPost results (contour design) ✓

2.4.3. Design from PLDesign directly (strip based region) ✓

2.4.4 Design from PLDesign directly (basic and additional reinforcement) ✓

2.5 Check deflections of slab

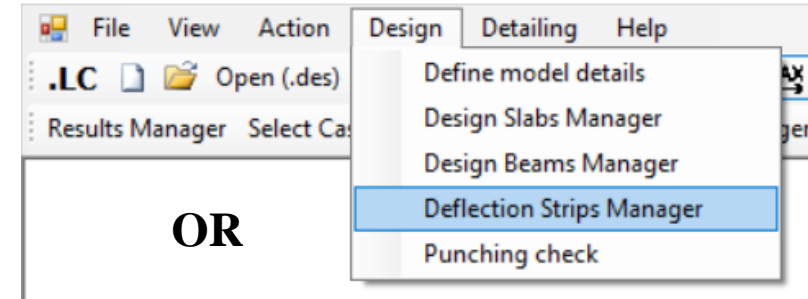
2.6 Check punching

2.7 Beam design

5 Check deflections of slab

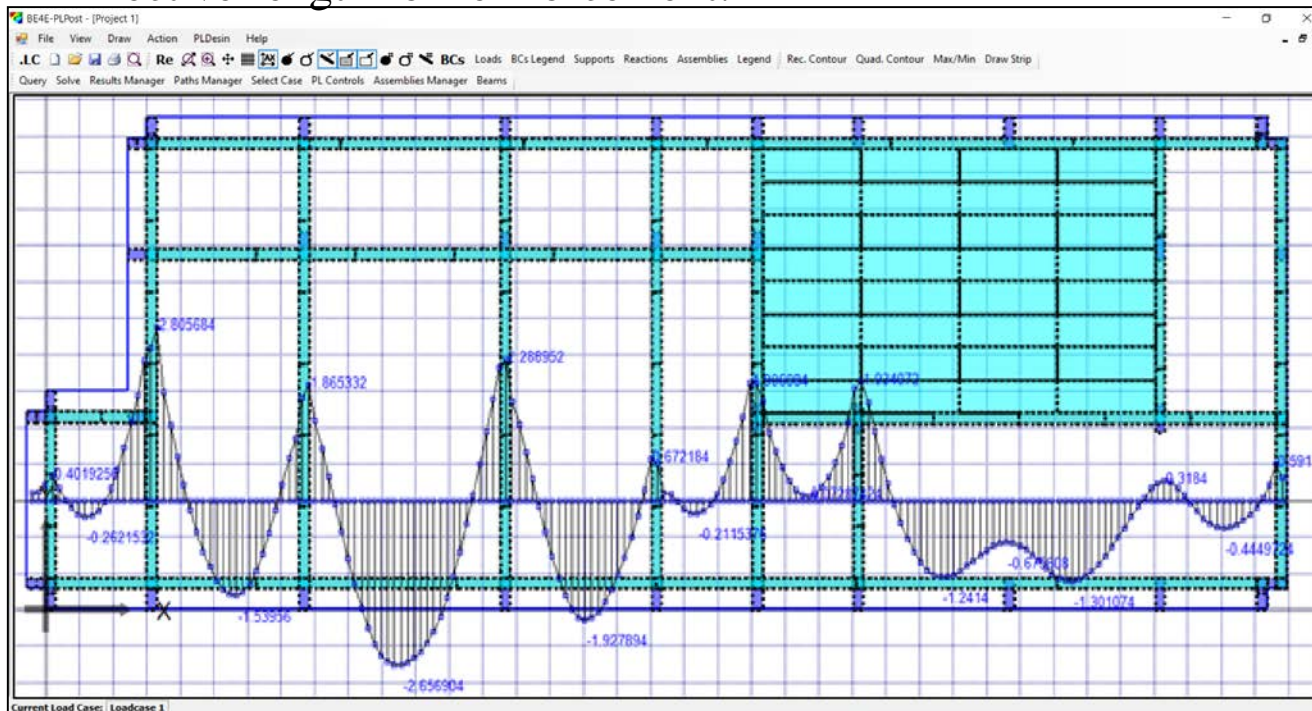
- The user can check deflection only for striped slab either from PLDesign Directly or import PLPost results.
- Open Slab deflection strip manager then load the strips needed to be checked.
- Choose the Major design parameter to calculate both of Maximum deflection and Effective length for reinforcement.

BE4E-PLDesign - [1.DL-title1-DL]



OR

Define model details Design Slabs Design Beams **Deflection Strips** Match properties Start detailing



Deflection Strips Manager

Deflection strips list:

Strip 1
 Strip 2

Add strips

Remove

Moment based.

Major design parameter:

Mxx

Maximum deflection:

-0.00076619000174105167

Effective length:

8.1077047545334384

Close

6 Check punching

- The user can check punching for column.
- Open assemblies manager and load .asm file.
- Add the required assemblies to be checked.
- Solve critical sections.
- Check punching.

www.be4e.com

Add assemblies

Support assemblies: Load assemblies:

1 2 3 4 5 6 7 8 9 10 11

Add Cancel

Assembly Manager

Support cells

Add

1 2 3 4 5 6 7 8 9 10

Assemblycells:

1,

ID: 1

Area: 0.15000009536743

CGIs User Defined:

CGOrdered Pair: 13.295,10.83

Inertia X: 0.00450000858975

Inertia Y: 0.00078125050775

Fz: 0

Mx: 0

My: 0

Inertia XL: 0.00450000858975

Inertia XYL: 8.59473039613312

Inertia YL: 0.00078125050775

Fz: 0

Mx L: 0

My L: 0

Theta: 0

Export

Use Principal

Load cells

Assemblycells:

ID:

Fz:

Mx:

My:

Load

Export All

OK

Punching assemblies

Punching assemblies

Support:1

Add Remove

Critical section properties

a: 0.25 Draw primary critical section

Beta: 2.40000152587890 Beta user defined.

b1: 0.25 Draw b1

b2: 0.60000038146972 Draw b2

Bo: 2.30000078678131 Bo user defined.

d: 0.15000000596046 Reset properties

Alpha: 40 Refresh

Column condition: Interior

BE analysis properties

Solve BE solution.

Material: Default Kgf-n

Status: ToBeSolved

Load case /combination: DL

Envelope design.

Envelope:

Spacing for BE solution: 0.1

Distance of secondary critical section: 2

Draw secondary critical section

Special items for EC design:

Axial stress in concrete: 0

Reinforcement ratio in dir-1: 0

Reinforcement ratio in dir-2: 0

Concrete shear capacity: 0

Solve critical sections

Check punching

Unbalanced critical shear stresses

Critical Shear stress: 0

Capacity ratio: NaN

UNSAFE

BE critical shear stresses

Reduction factor for non-linearity effect in BE-results: 0.15

Critical Shear stress: 0

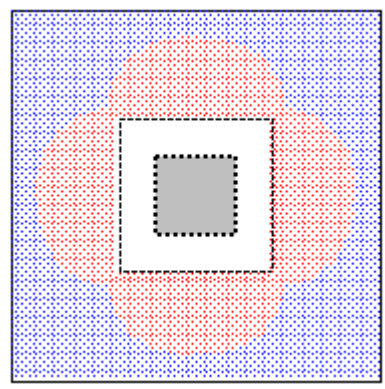
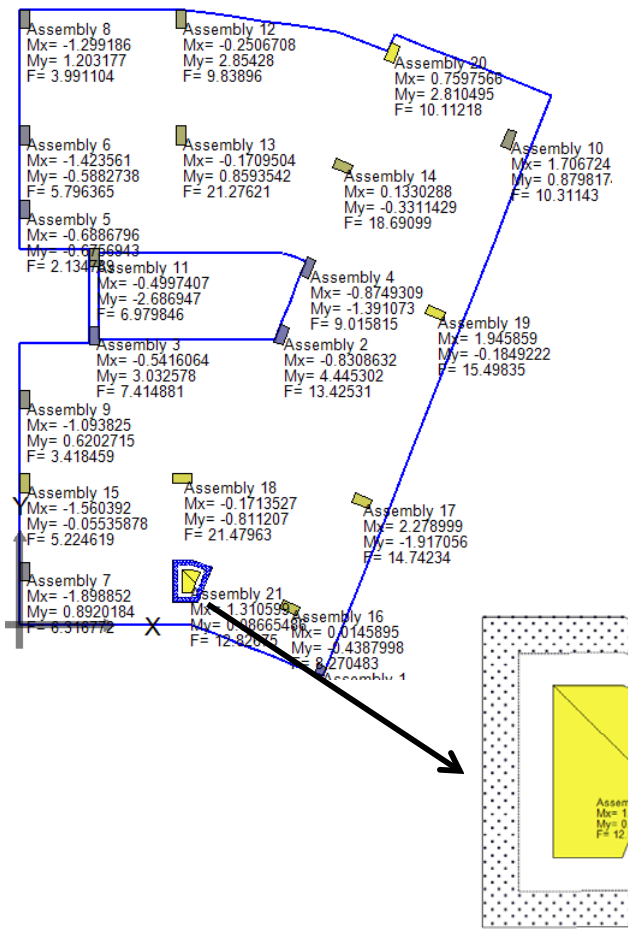
Capacity ratio: NaN

UNSAFE

Close

6 Check punching

Punching Check According to ECP 2003 and ACI



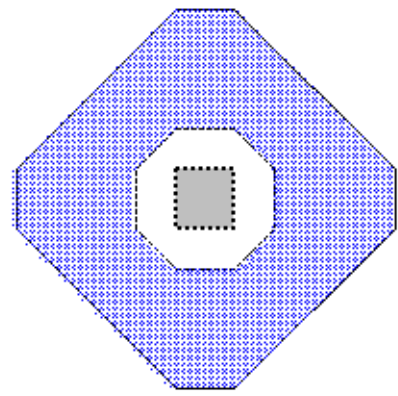
Punching assemblies

Punching assemblies	Critical section properties	BE analysis properties	Special items for EC design:	Unbalanced critical shear stresses
Support:1	a: 1.10000038146972 b: 1.10000038146972 Beta: 1 b1: 1.10000038146972 b2: 1.10000038146972 Bo: 8.40000152587890 d: 1 Alpha: 40 Column condition: Interior	<input checked="" type="checkbox"/> Solve BE solution. Material: Default Tonf Status: ToBeSolved Load case /combination: LoadCase 1 <input type="checkbox"/> Envelope design. Envelope: Spacing for BE solution: 0.1 Distance of secondary critical section: 2	Axial stress in concrete: 0 Reinforcement ratio in dir-1: 0 Reinforcement ratio in dir-2: 0 Concrete shear capacity: 130.272011680	Critical Shear stress: 3.41515860566 Capacity ratio: 2.62155973614 SAFE BE critical shear stresses Reduction factor for non-linearity effect in BE-results: 0.15 Critical Shear stress: 272.760740661 Capacity ratio: 2.09377852651 UNSAFE

Punching Check According to EC

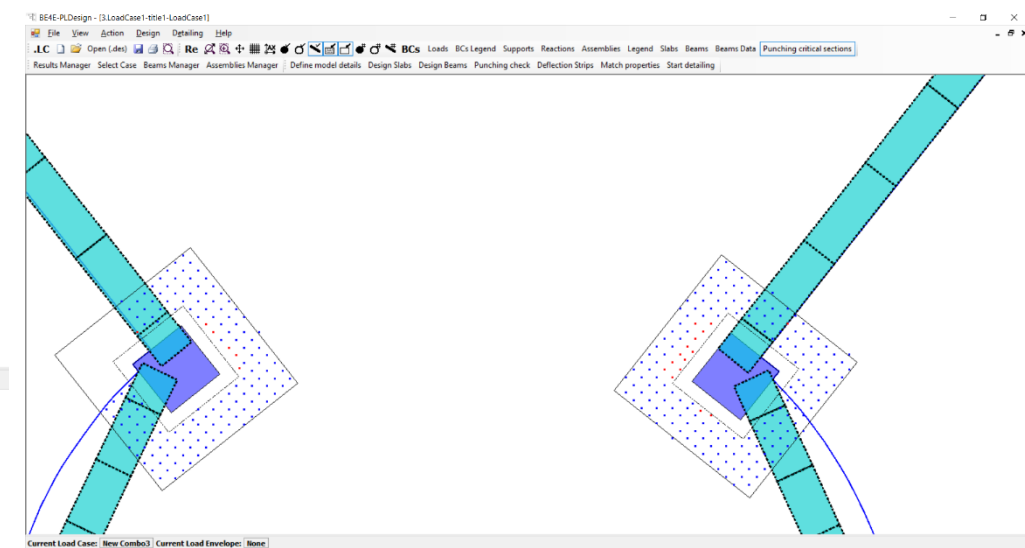
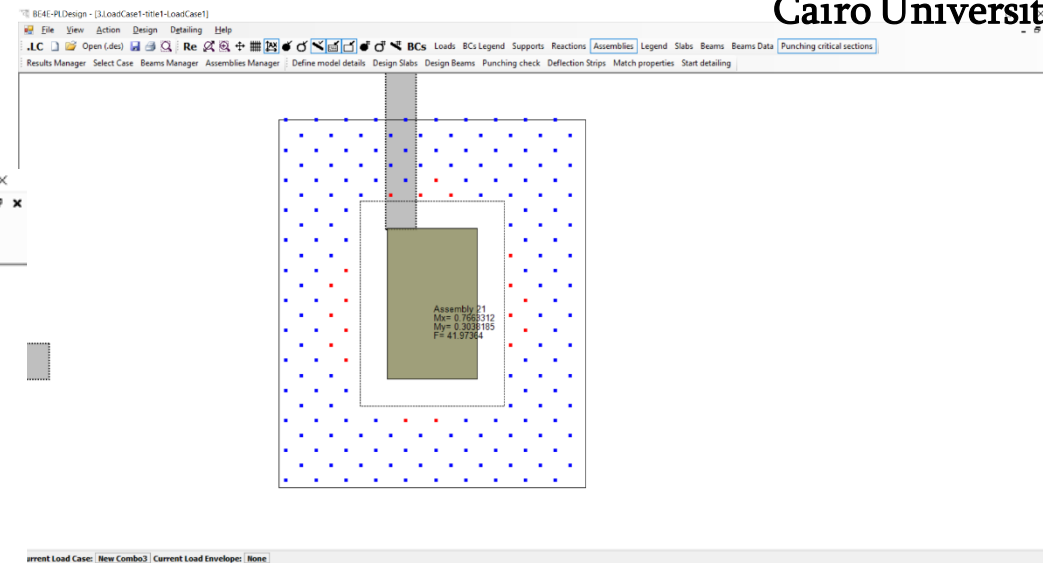
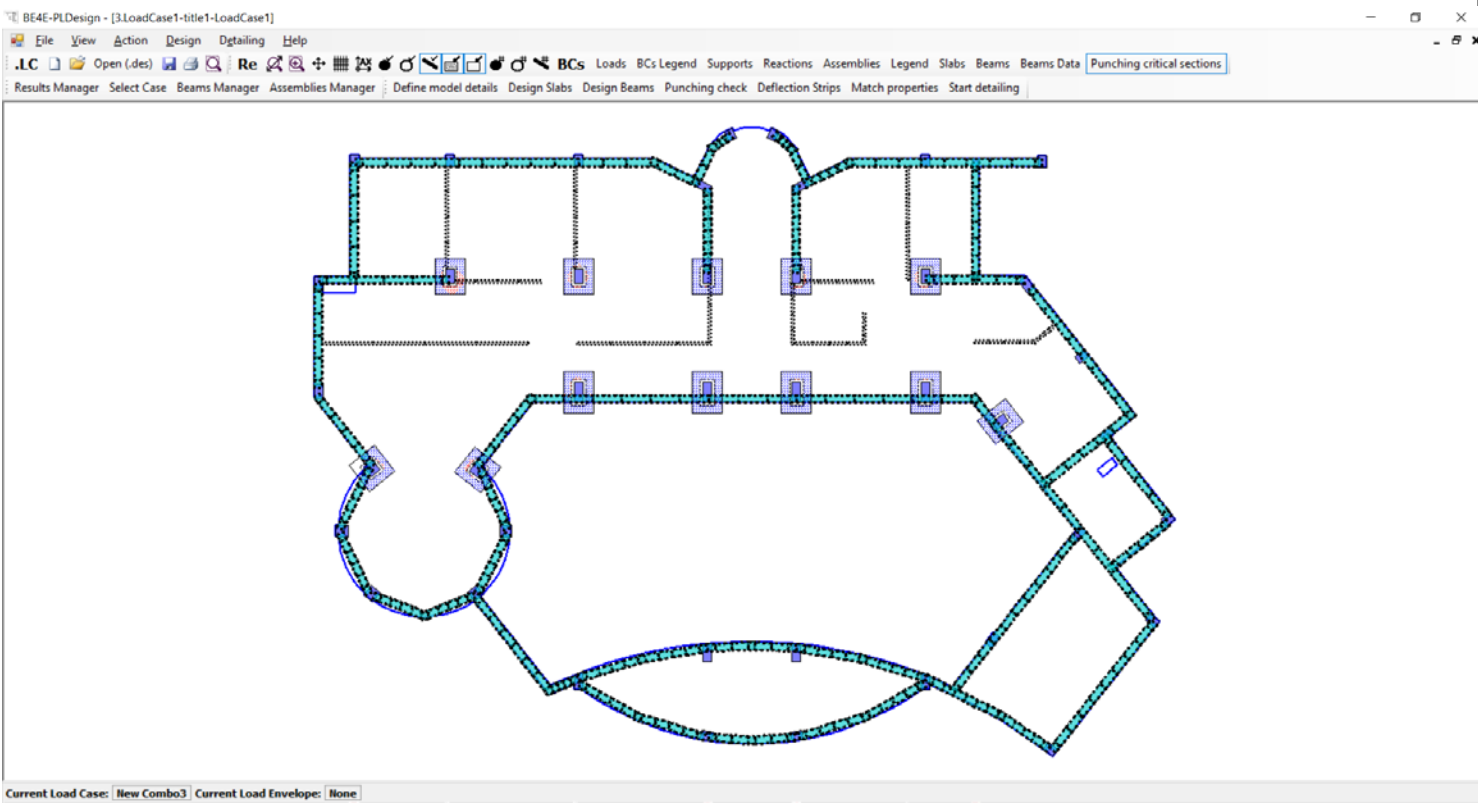
Punching assemblies

Punching assemblies	Critical section properties	BE analysis properties	Special items for EC design:	Unbalanced critical shear stresses
Support:1	a: 1.10000038146972 b: 1.10000038146972 Beta: 1 b1: 1.10000038146972 b2: 1.10000038146972 Bo: 10.4000015258789 d: 1.5 Alpha: 40 Column condition: Interior	<input checked="" type="checkbox"/> Solve BE solution. Material: Default Tonf Status: ToBeSolved Load case /combination: LoadCase 1 <input type="checkbox"/> Envelope design. Envelope: Spacing for BE solution: 0.1 Distance of secondary critical section: 2	Axial stress in concrete: 0 Reinforcement ratio in dir-1: 0 Reinforcement ratio in dir-2: 0 Concrete shear capacity: 28.1869116360	Critical Shear stress: 1.92545218976 Capacity ratio: 6.8301509887 SAFE BE critical shear stresses Reduction factor for non-linearity effect in BE-results: 0.15 Critical Shear stress: 19.2031991958 Capacity ratio: 0.68128071084 SAFE



6 Check punching

www.be4e.com



PLDesign Package

2.1. File needed to be exported before using PLDesign ✓

2.2. Starting PLDesign ✓

2.3. Load combinations & load envelopes ✓

2.4. Slab design ✓

2.4.1. Design from PLPost results (strip design) ✓

2.4.2. Design from PLPost results (contour design) ✓

2.4.3. Design from PLDesign directly (strip based region) ✓

2.4.4 Design from PLDesign directly (basic and additional reinforcement) ✓

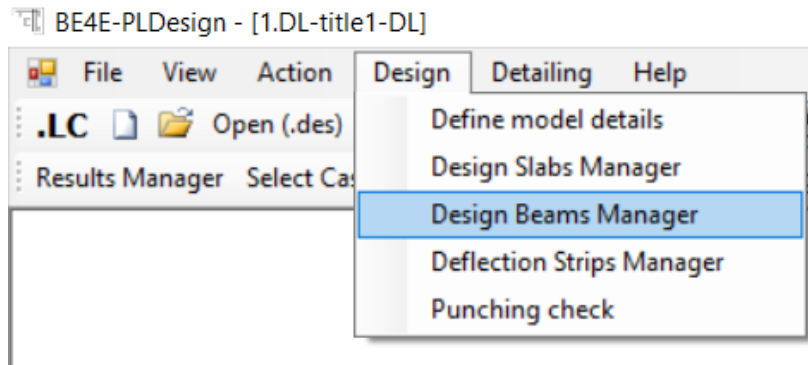
2.5 Check deflections of slab ✓

2.6 Check punching ✓

2.7 Beam design

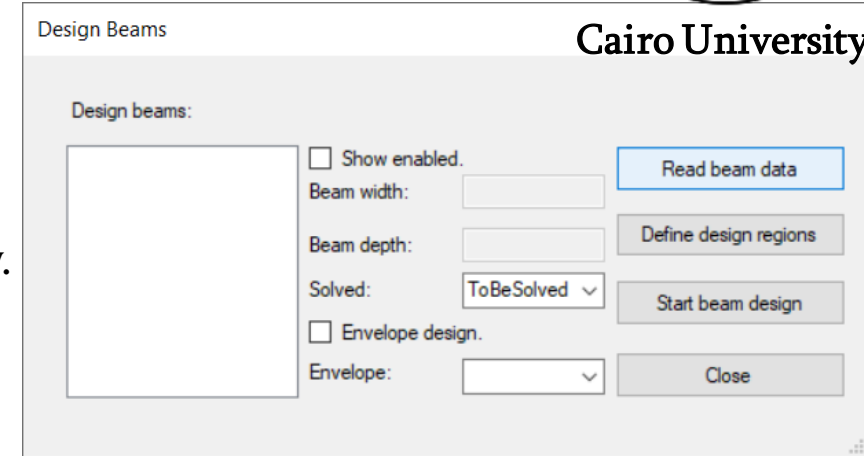
7 Beam design

- In PLDesign the user can design or check design on beam reinforcement.
- But the user should export beam assemblies from Gen file to read beam geometry.
- Open PLDesign Manager then load beam data (.basm) to read beams geometry.
- Now the user should define the design regions in beams.

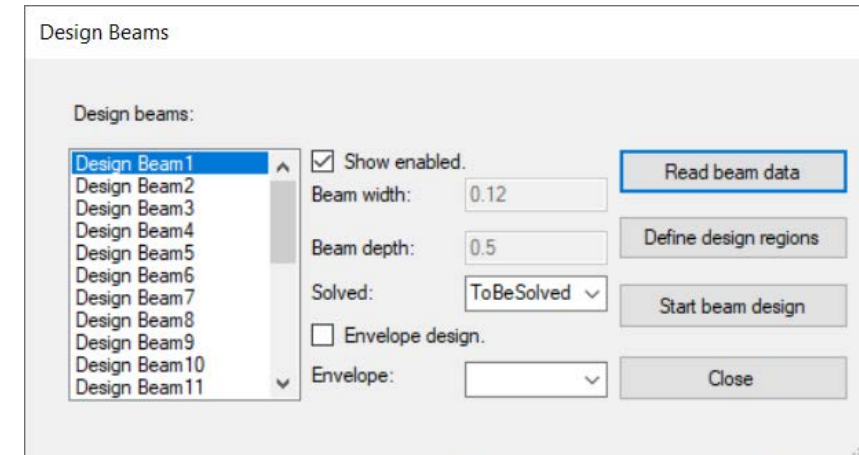


OR

Define model details Design Slabs **Design Beams** Deflection Strips Match properties Start detailing



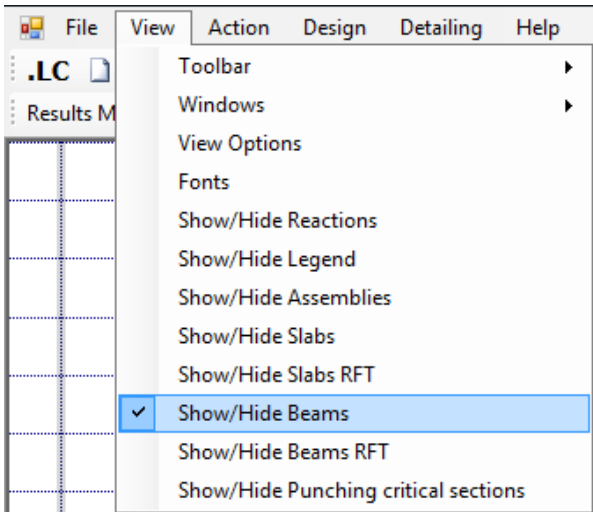
Before loading beam assembly file.



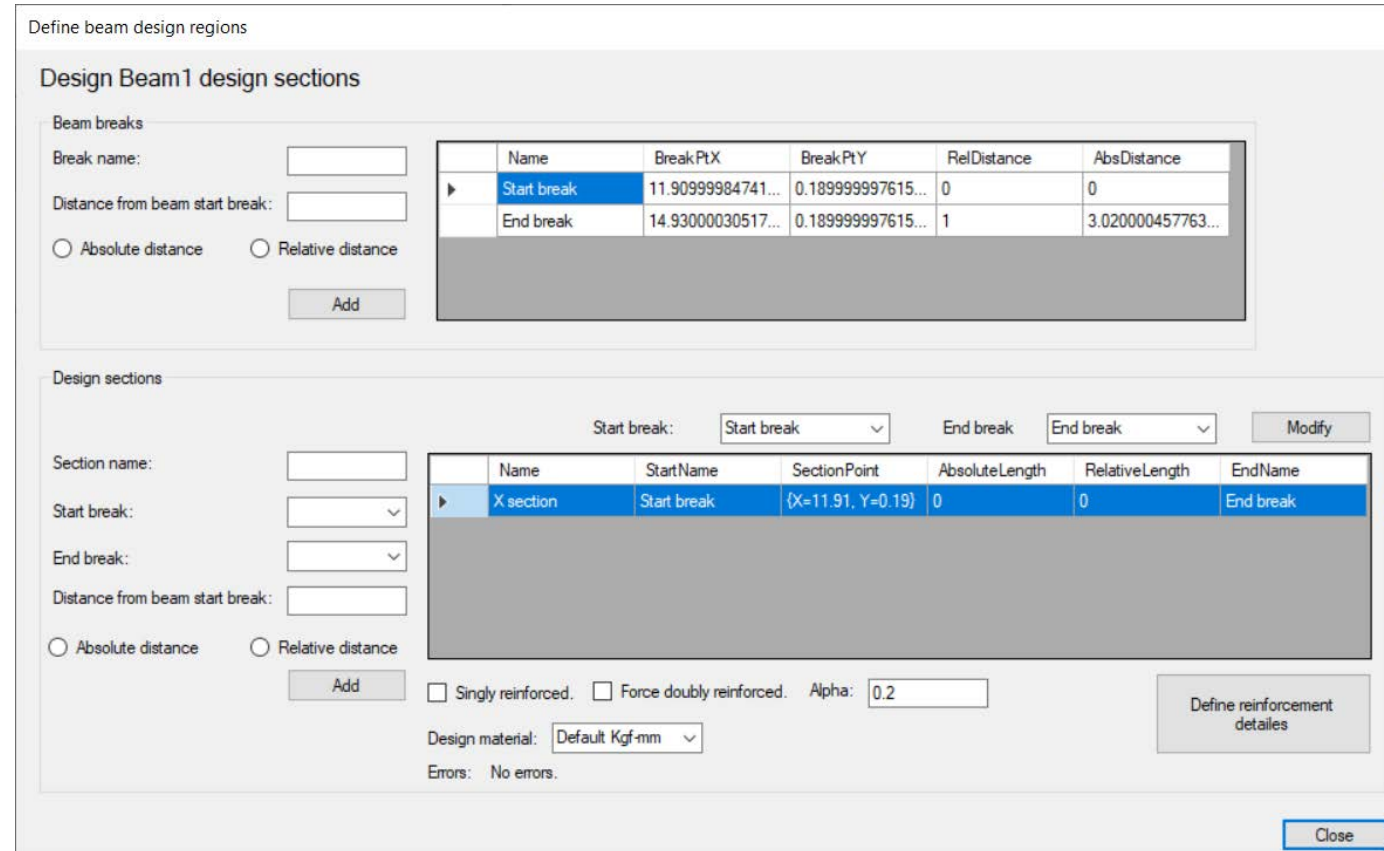
After loading beam assembly file.

7 Beam design

- Open Define design region to show the design beam section.
- The user will notice that the design beam is divided into two parts, the first part is for the beam breaks (supports) and the other is for beam sections.
- Where is the beam that the user design?

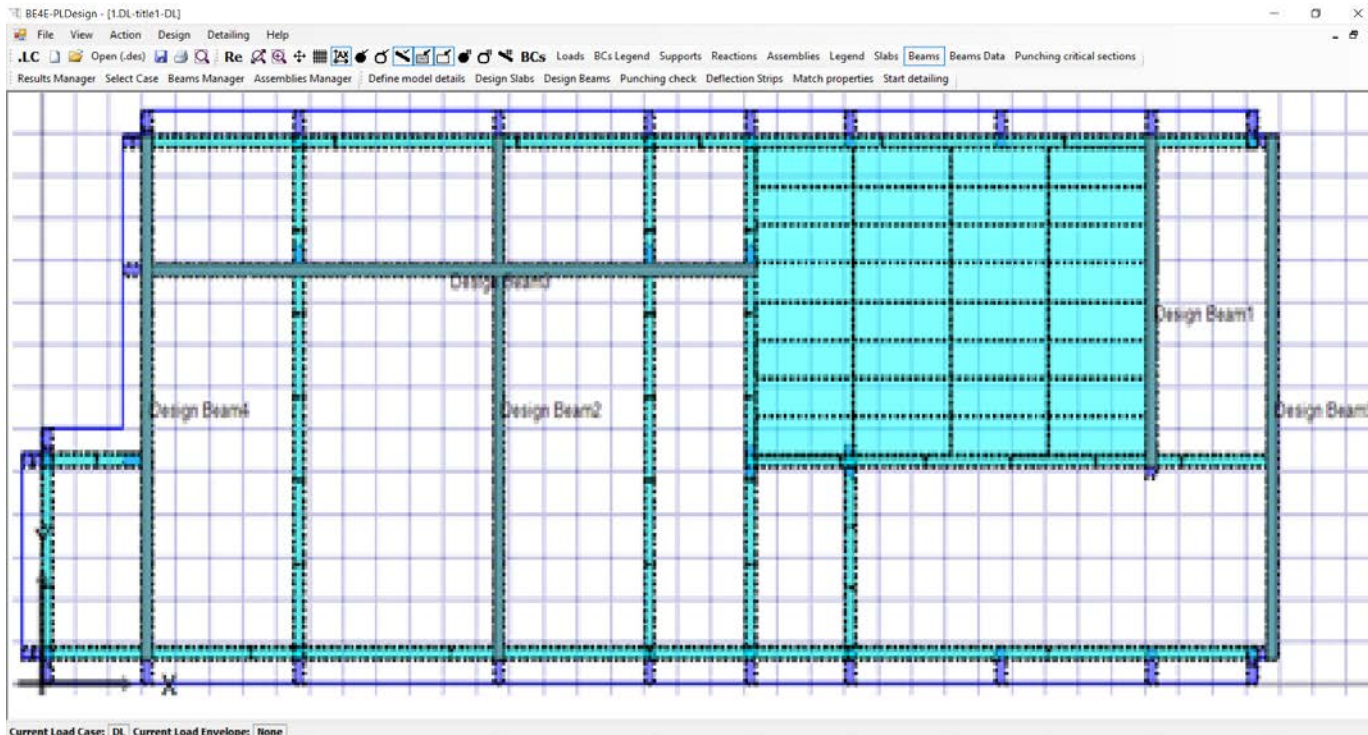


OR



7 Beam design

- Show/hide beams allow the user to see the beam to determine the section he needs.
- Every beam contain mark box (Show enabled), the default is all beams are enabled, but the user can open or close any beam he want.
- Once the user know the location of beams need for design, he could determine the sections and breaks.



7 Beam design

- The user should connect between the AutoCAD drawing & the PLDesign to put the section accurately.
- In beam 1 contain 3 supports so the user should put section break and five sections (Two sections +ve and three sections -ve).

Define beam design regions

Design Beam1 design sections

Beam breaks

Break name:

Distance from beam start break:

Absolute distance Relative distance

Name	BreakPTX	BreakPTY	RelDistance	AbsDistance
Start break	33.1800079345...	12.9300498962...	0	0
End break	33.1800079345...	5.10004901885...	1	7.83000087738...
1	33.1800079345...	10.1253435819...	0.35819999999...	2.80470631427...

Design sections

Start break: End break:

Section name:

Start break:

End break:

Distance from beam start break:

Absolute distance Relative distance

Name	StartName	SectionPoint	AbsoluteLength	RelativeLength	EndName
A	Start break	{X=33.18001, Y...	1.40235328674...	0.50000004620...	1
B	1	{X=33.18001, Y...	2.51264741115...	0.50000002579...	End break
-A	Start break	{X=33.18001, Y...	0	0	1
-B	1	{X=33.18001, Y...	2.59208679409...	5.15807931564...	End break
-C	1	{X=33.18001, Y...	5.02529456310...	1	End break

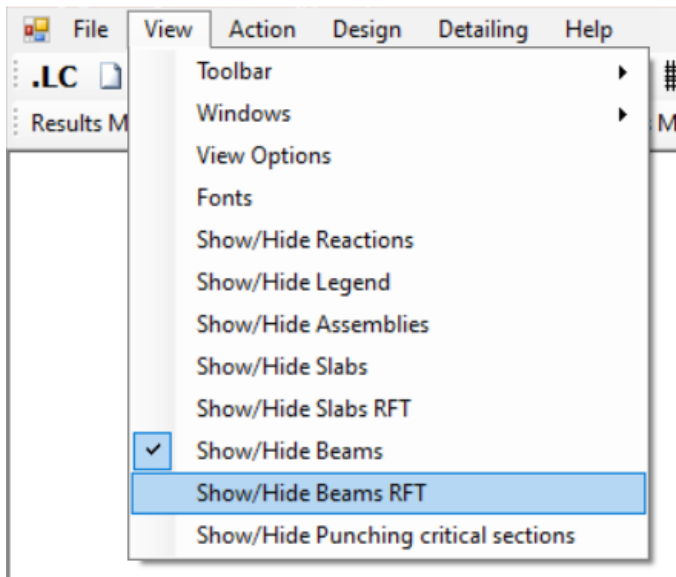
Singly reinforced. Force doubly reinforced. Alpha:

Design material: Errors: No errors.

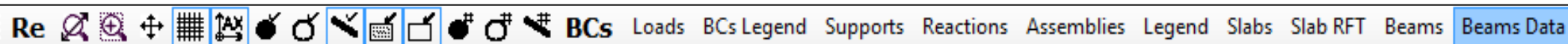
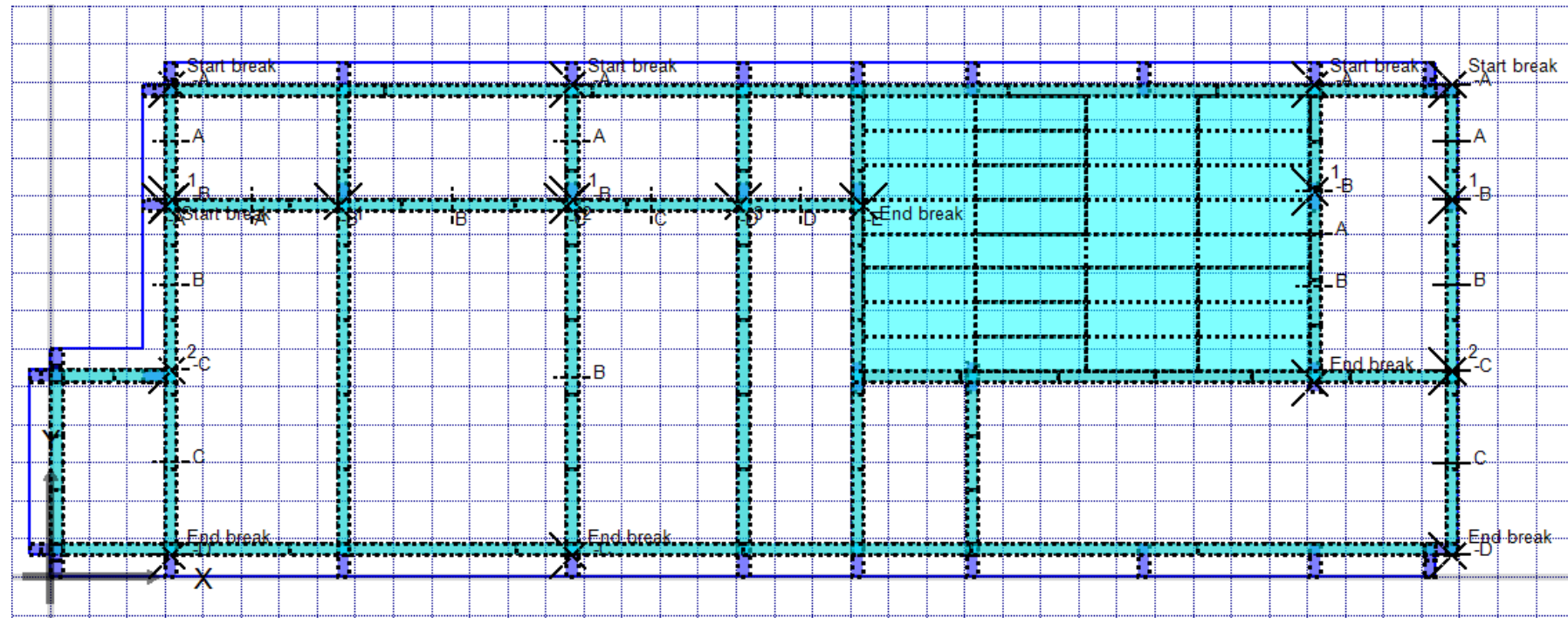
7 Beam design

- The user can check the location of sections that he constructed.

BE4E-PLDesign - [1.DL-title1-DL]



OR



7 Beam design

- Open Define reinforcement details for +ve section and another for -ve section.

Define reinforcement details

Design Beam 1 - A reinforcement details

Straining actions

Design cell: 52 Design beam element: 4

Flexure design load case/combination: ultimate Design moment: 12.7224931716

Shear design load case/combination: ultimate Design shear: 13.7844972610

Torsion design load case/combination: ultimate Design torsion: 0.53598266839

Reinforcement layers

NoOfBars	BarDiameter	depth	IsBottomLayer
4	0.016	0	<input checked="" type="checkbox"/>
2	0.016	0.032	<input checked="" type="checkbox"/>
2	0.016	0	<input type="checkbox"/>

Required Asteel top: 0 Actual Asteel top: 0.00040212385

Required Asteel bottom: 0.00085611111 Actual Asteel bottom: 0.00120637157

Bending bottom: Safe Add reinforcement layer

Bending top: Safe

Stirrups

Width	NoOfLegs	BarDiameter	BarSpacing
0.24	2	0.008	0.1

Required Asteel: 0.00087677385 Actual Asteel: 0.00100530964

Stirrups (shear+torsion): Safe Add stirrup

Longitudinal rebars

BarDiameter	Xbar	Ybar
0.016	0.04	0.25
0.016	0.04	0.5
0.016	0.26	0.25
0.016	0.26	0.5

Required Asteel: 0.00070116111 Actual Asteel: 0.00080424771

Longitudinal bars (torsion): Safe Add longitudinal bar

Cover top: 0.025

Cover left: 0.025

Cover right: 0.025

Cover bottom: 0.025 Refresh

Close

Define reinforcement details

Design Beam 1 - B reinforcement details

Straining actions

Design cell: 51 Design beam element: 3

Flexure design load case/combination: ultimate Design moment: -19.5662860870

Shear design load case/combination: ultimate Design shear: -15.2343101501

Torsion design load case/combination: ultimate Design torsion: 0.21970474720

Reinforcement layers

NoOfBars	BarDiameter	depth	IsBottomLayer
4	0.016	0	<input type="checkbox"/>
2	0.016	0.032	<input type="checkbox"/>
2	0.016	0	<input checked="" type="checkbox"/>

Required Asteel top: 0.00085611111 Actual Asteel top: 0.00201061929

Required Asteel bottom: 0 Actual Asteel bottom: 0.00040212385

Bending bottom: Safe Add reinforcement layer

Bending top: Safe

Stirrups

Width	NoOfLegs	BarDiameter	BarSpacing
0.24	2	0.008	0.1

Required Asteel: 0.00087677385 Actual Asteel: 0.00100530964

Stirrups (shear+torsion): Safe Add stirrup

Longitudinal rebars

BarDiameter	Xbar	Ybar
0.016	0.04	0.25
0.016	0.04	0.5
0.016	0.26	0.25
0.016	0.26	0.5

Required Asteel: 0.00070116111 Actual Asteel: 0.00080424771

Longitudinal bars (torsion): Safe Add longitudinal bar

Cover top: 0.025

Cover left: 0.025

Cover right: 0.025

Cover bottom: 0.025 Refresh

Close

7 Beam design

- Match the reinforcement properties for all +ve/-ve sections.
- Check that the sections have been matched it's properties.

Match properties

Slab spans | Beams | **Beam sections** | Punching asms.

Source beam : Design Beam1
Destination beam: Design Beam4

Source section : A, B, -A, -B, -C
Destination sections: A, B, C, -A, -B, -C, -D

Design data
 Design materials
 Bending moment Loadcase
 Shear Loadcase
 Torsion Loadcase

Reinforcement data
 Layers (flexure)
 Stirrups
 Longitudinal bars

Dimensions
 Bottom cover
 Top cover
 Left cover
 Right cover

Section data
 Is Singly reinforced
 Force doubly reinforced section.
 Alpha values

Original
 Add to destination data
 Replace destination data

Match sections

Close

Match properties

Slab spans | Beams | **Beam sections** | Punching asms.

Source beam : Design Beam1
Destination beam: Design Beam5

Source section : A, B, -A, -B, -C
Destination sections: A, B, C, -A, -B, -C, -D

Design data
 Design materials
 Bending moment Loadcase
 Shear Loadcase
 Torsion Loadcase

Reinforcement data
 Layers (flexure)
 Stirrups
 Longitudinal bars

Dimensions
 Bottom cover
 Top cover
 Left cover
 Right cover

Section data
 Is Singly reinforced
 Force doubly reinforced section.
 Alpha values

Original
 Add to destination data
 Replace destination data

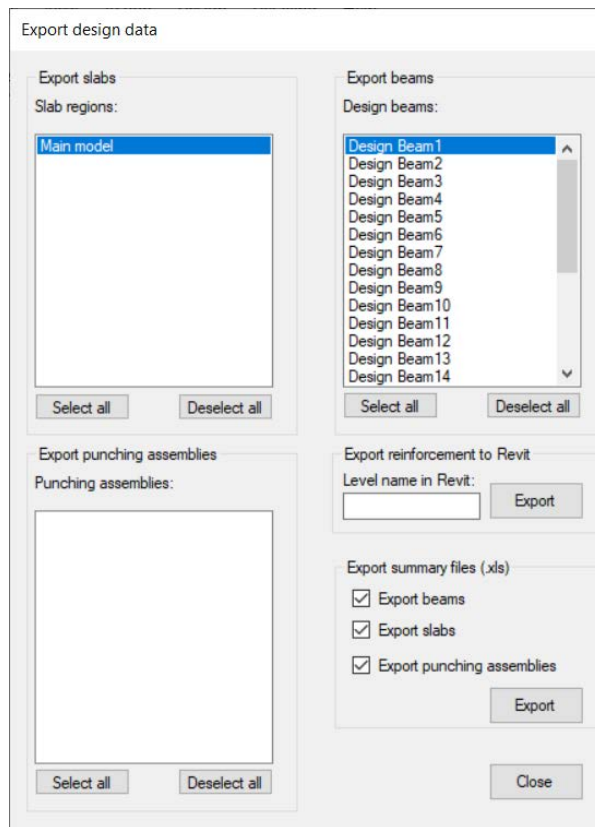
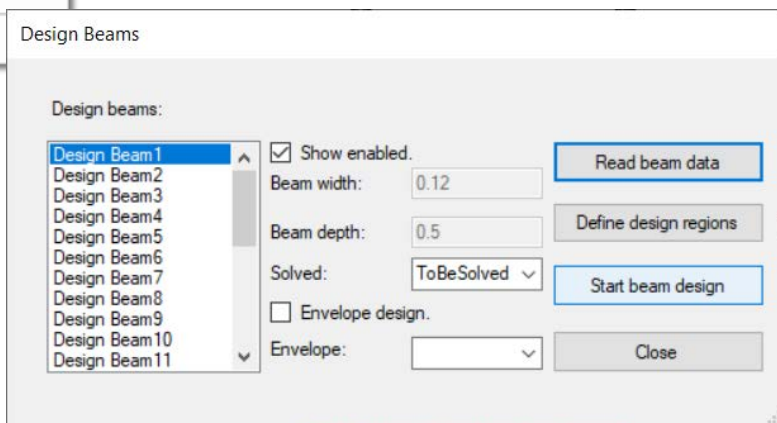
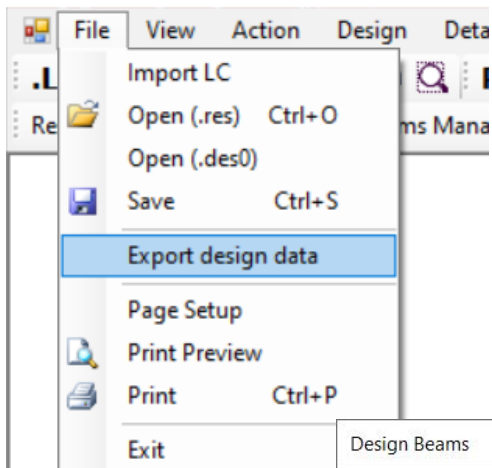
Match sections

Close

7 Beam design

- Start beam design then check the reinforcement that the user insert.

BE4E-PLDesign - [1.DL-title1-DL]

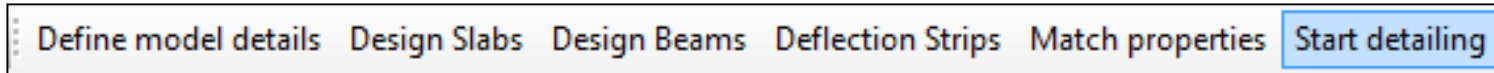


Company Name: _____
 Project Name: _____
 Designed By: _____
 Reviewed By: _____
 Approved by: _____

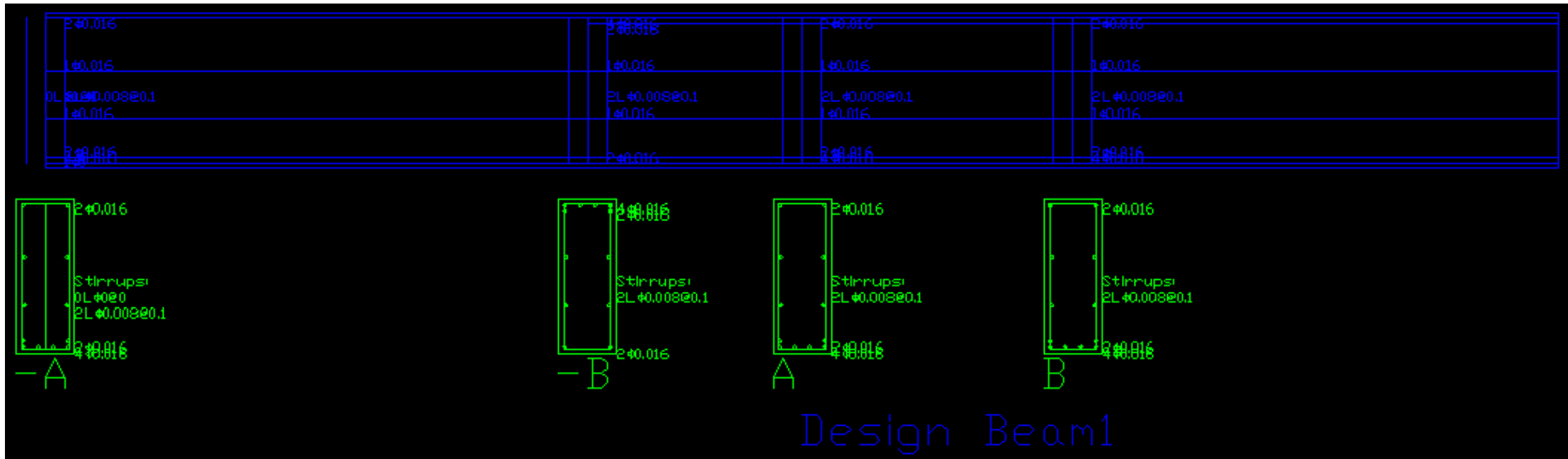
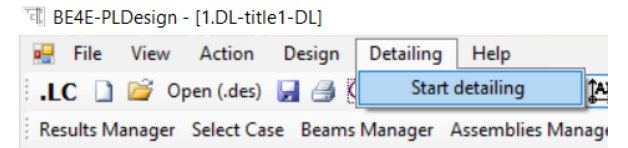
Beam name	Beam size	Beam section	Flexure reinforcement	Stirrups	Longitudinal steel
Design Beam1	0.3 X 0.8	A	Top (2 Φ 0.016) Bot (4 Φ 0.0162 Φ 0.016)	2L Φ 0.008 @ 0.1	Φ 0.016 @ (0.04,0.25) Φ 0.016 @ (0.04,0.5) Φ 0.016 @ (0.26,0.25) Φ 0.016 @ (0.26,0.5)
		B	Top () Bot ()		
		-A	Top () Bot (4 Φ 0.016)	0L Φ 0 @ 0	Φ 0 @ (0,0)
		-B	Top (4 Φ 0.0162 Φ 0.016) Bot (2 Φ 0.016)	2L Φ 0.008 @ 0.1	Φ 0.016 @ (0.04,0.25) Φ 0.016 @ (0.04,0.5) Φ 0.016 @ (0.26,0.25) Φ 0.016 @ (0.26,0.5)

7 Beam design

- Now the user are going to Start detailing for the beams.
- This detailing shows the beam reinforcement and the reinforcement of the sections.



OR



PLDesign Package

2.1. File needed to be exported before using PLDesign ✓

2.2. Starting PLDesign ✓

2.3. Load combinations & load envelopes ✓

2.4. Slab design ✓

2.4.1. Design from PLPost results (strip design) ✓

2.4.2. Design from PLPost results (contour design) ✓

2.4.3. Design from PLDesign directly (strip based region) ✓

2.4.4 Design from PLDesign directly (basic and additional reinforcement) ✓

2.5 Check deflections of slab ✓

2.6 Check punching ✓

2.7 Beam design ✓

CUFE-BE



www.be4e.com



Cairo University

- A. PLGen – Model generator
- B. PLView – Numerical model
- C. PLCoreMan – Manager and solver
- D. PLPost – Post processing
- E. PLDesign – Design tool
- F. PLPAK modelling capabilities**

F. PLPAK modelling capabilities

- Structural members in the PLPAK are entered with their exact dimensions (geometry) e.g. columns and beams are no longer entered as nodes or lines, but represented with their actual proportions thus no peaking occurs on slab region contour results.
- 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.
- 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.

F. PLPAK modelling capabilities

- Easy drawing capabilities of the PLPAK.
- The soil springs in soil models are considered as continuous spring patches underneath the continuum foundation plate.
- Exporting and importing model as text files.
- Real time post-processing.
- Strip results and exporting it to excel.
- Post-processing for certain part of the slab in huge practical problems.
- Direct assemblies reactions (walls / Cores).
- Real and easy assigning of assembly loads (walls / Cores).
- Save and restore the results in PLPost.

CUFE-BE



www.be4e.com



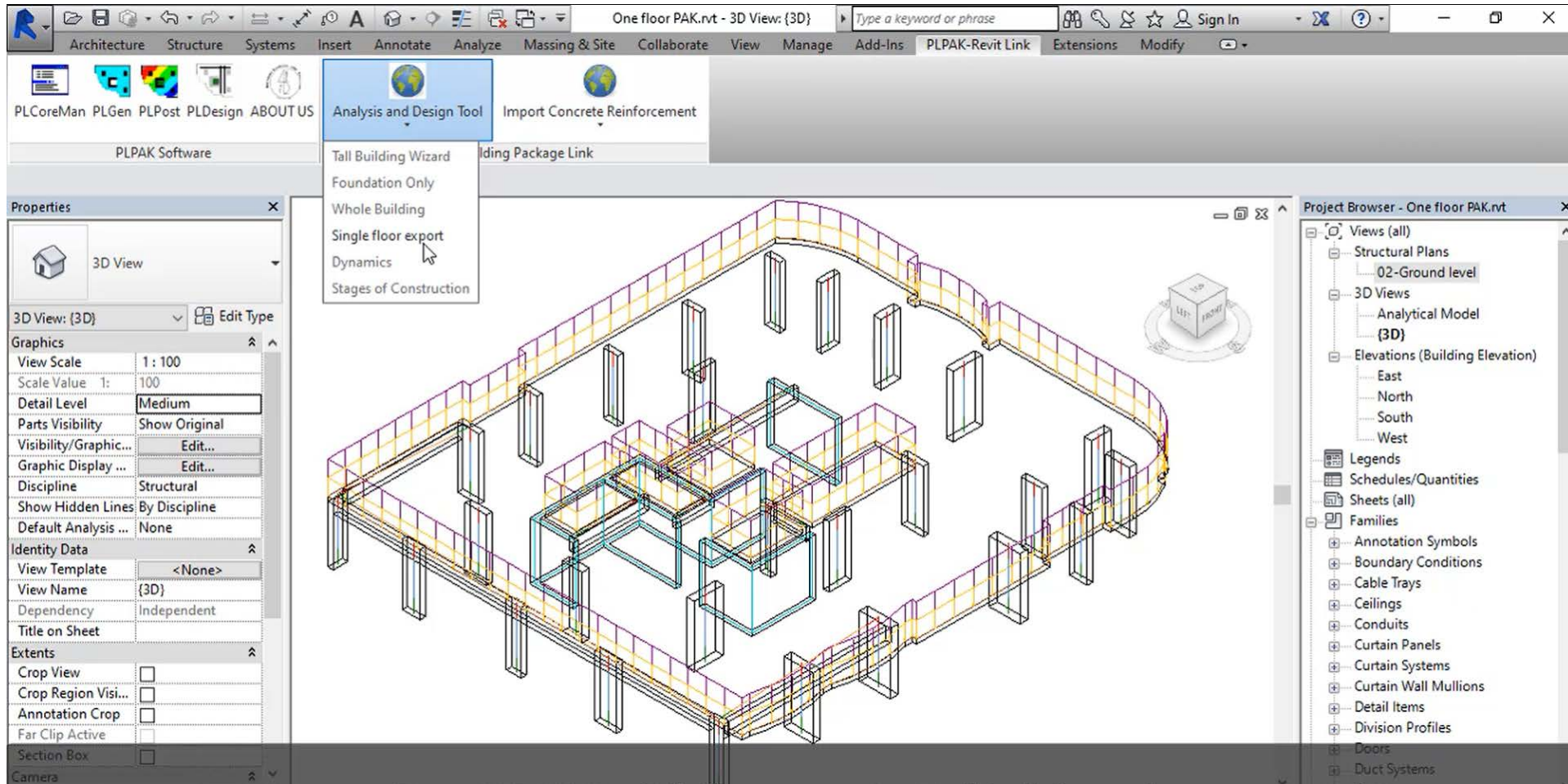
Cairo University

Table of contents

- 1. Introduction**
- 2. Basic package (PLPAK Basic)**
- 3. One floor package (BIM-PLPAK)**
4. Foundation Package (FoundPAK)
5. Fixed base Package (FBPAK)
6. Post-tension Package (PTPAK)
7. Dynamic package (DynPAK)
8. Overall building package (OBPAK)
9. 4D and 5D analysis
10. Conclusions

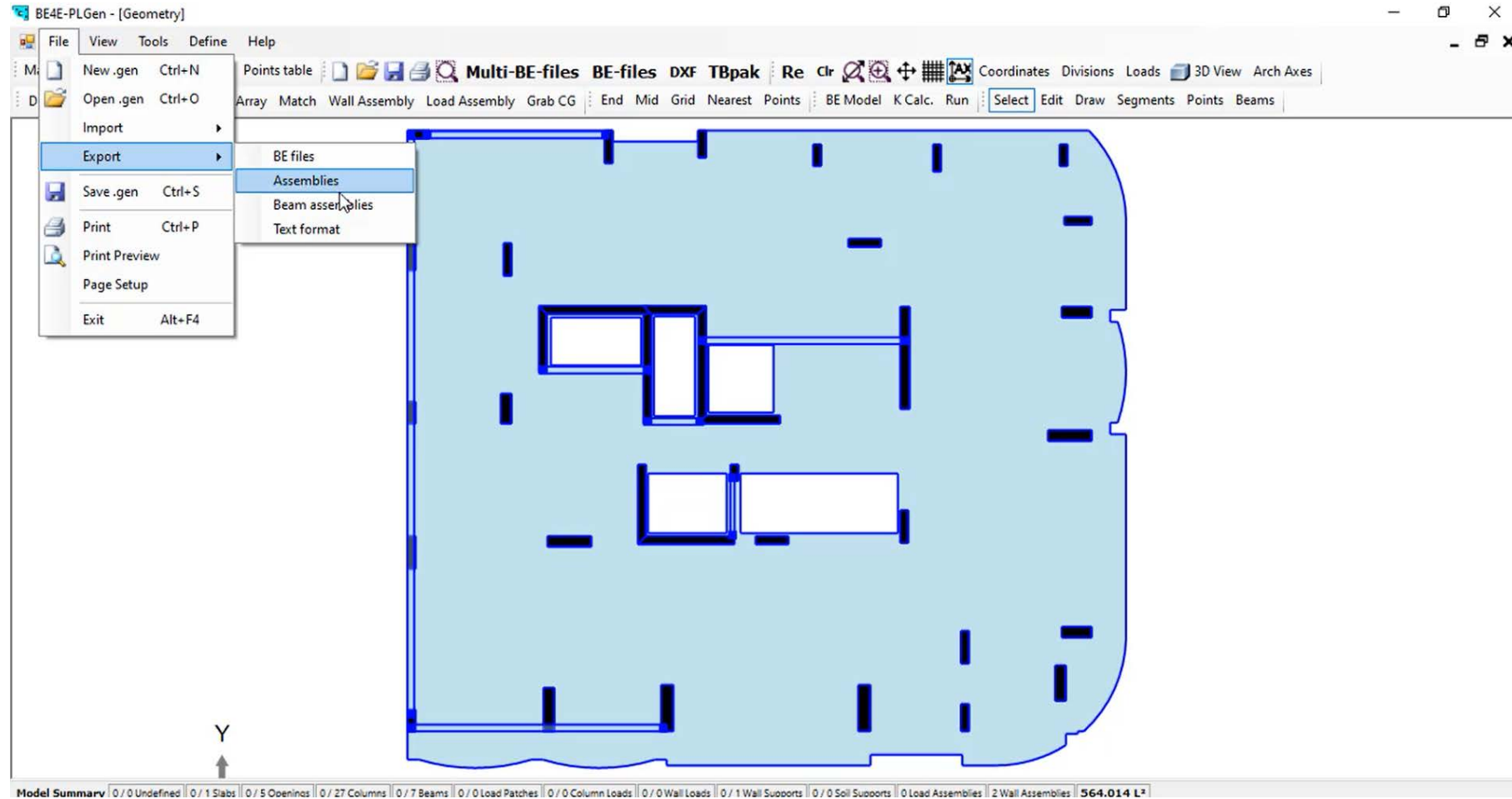
3. One floor package (BIM-PLPAK)

www.be4e.com

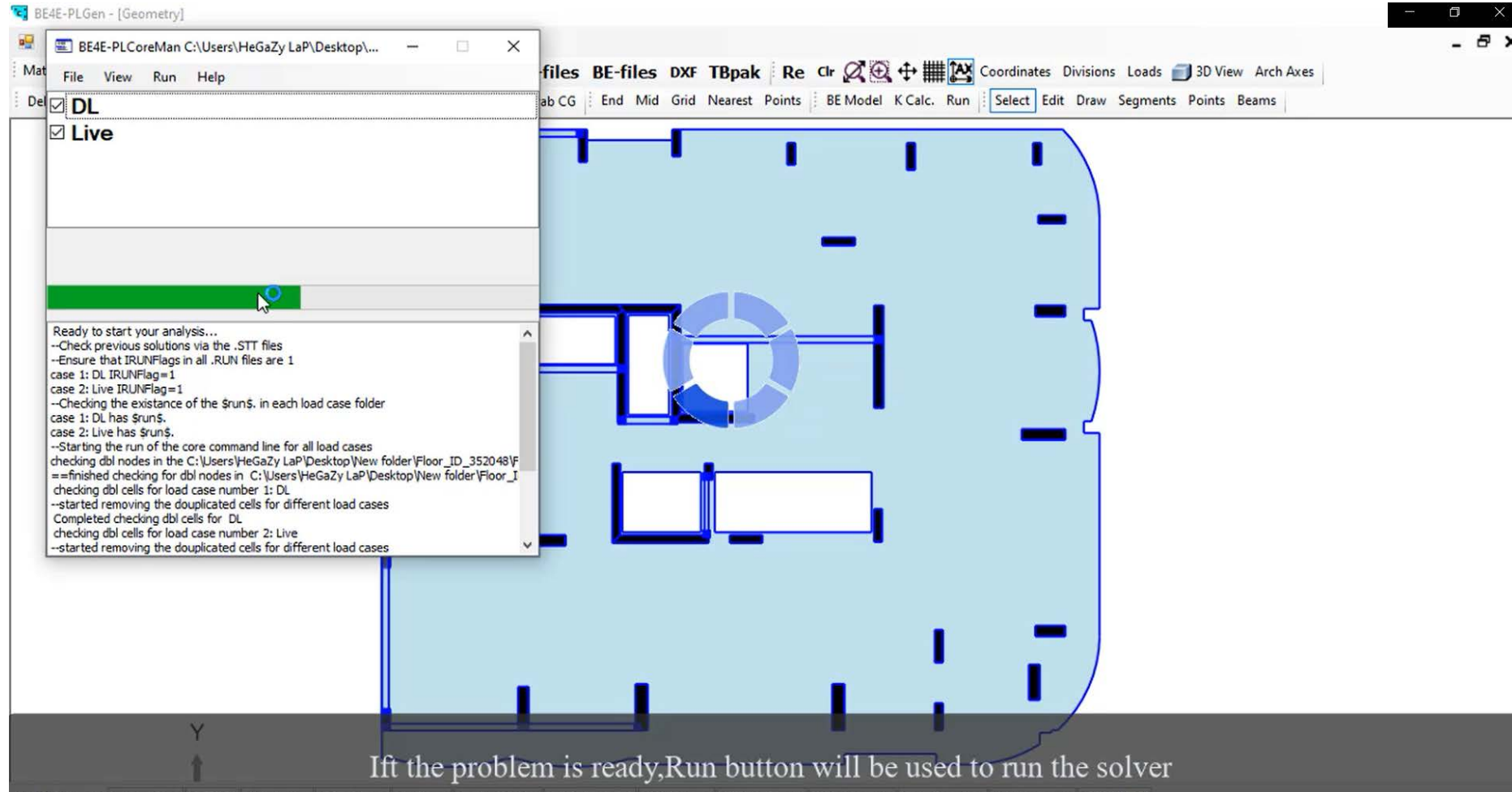


3. One floor package (BIM-PLPAK)

www.be4e.com



3. One floor package (BIM-PLPAK)



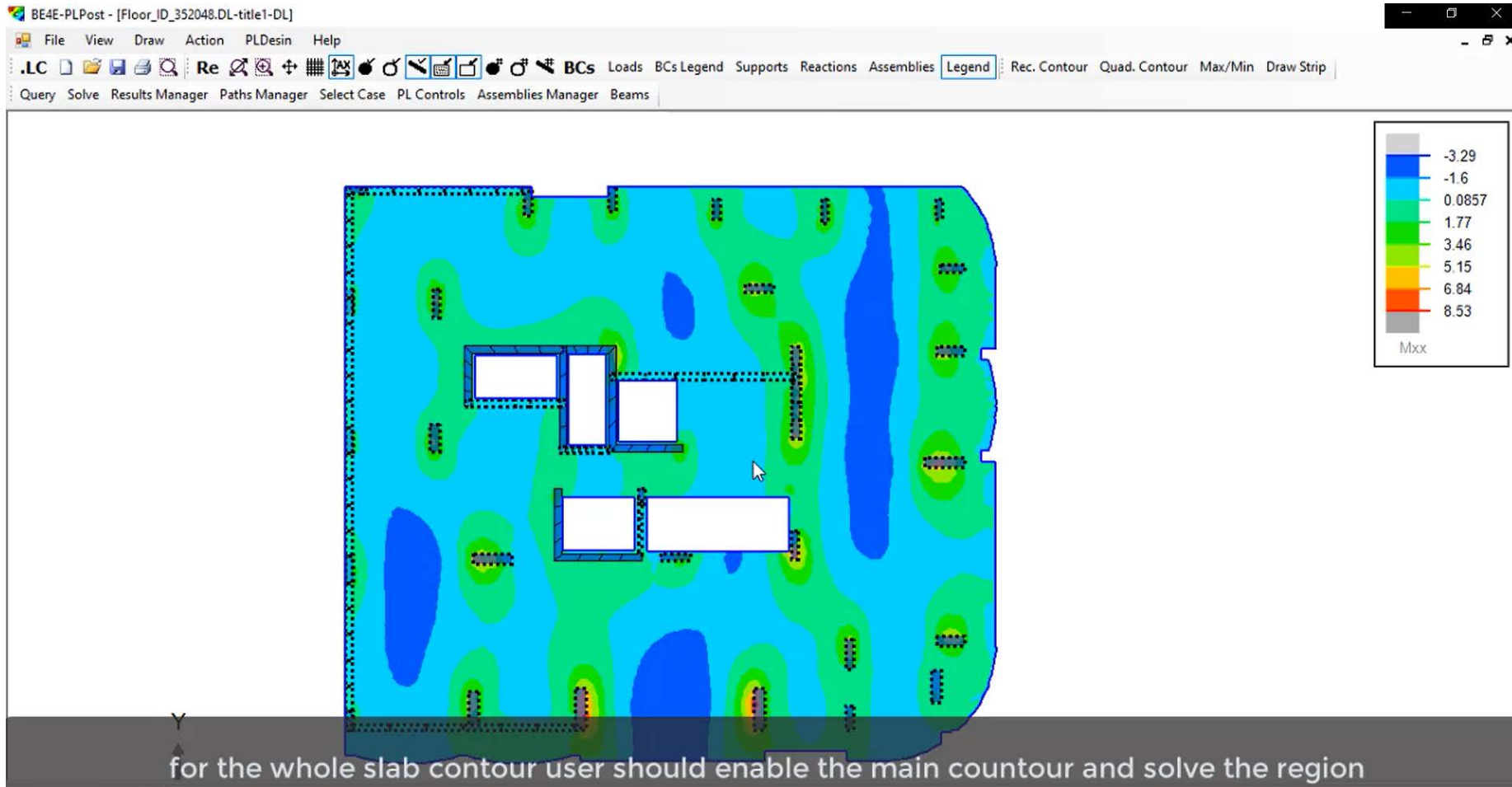
The screenshot displays the BE4E-PLGen software interface. The main window shows a 2D floor plan model with a central circular structure and various rooms. A console window is open, displaying the following text:

```
Ready to start your analysis...  
--Check previous solutions via the .STT files  
--Ensure that IRUNFlags in all .RUN files are 1  
case 1: DL IRUNFlag=1  
case 2: Live IRUNFlag=1  
--Checking the existance of the $run$. in each load case folder  
case 1: DL has $run$.  
case 2: Live has $run$.  
--Starting the run of the core command line for all load cases  
checking dbl nodes in the C:\Users\HeGaZy LaP\Desktop\Floor_ID_352048\F  
==finished checking for dbl nodes in C:\Users\HeGaZy LaP\Desktop\Floor_I  
checking dbl cells for load case number 1: DL  
--started removing the douplicated cells for different load cases  
Completed checking dbl cells for DL  
checking dbl cells for load case number 2: Live  
--started removing the douplicated cells for different load cases
```

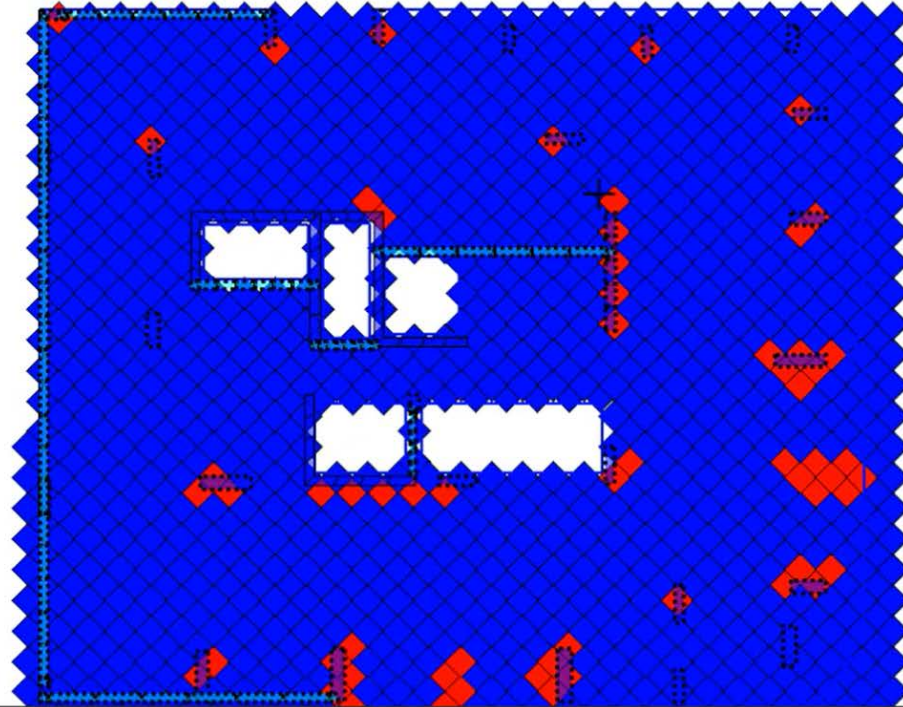
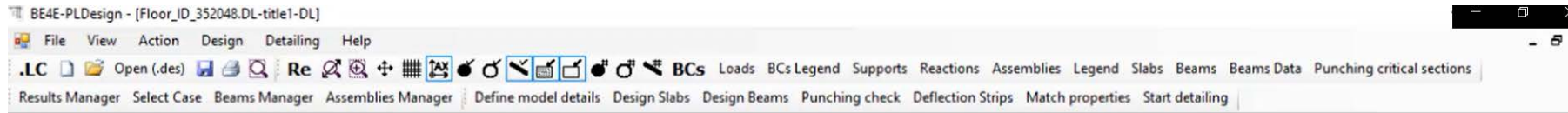
At the bottom of the screenshot, a text box contains the instruction: "If the problem is ready, Run button will be used to run the solver".

3. One floor package (BIM-PLPAK)

www.be4e.com

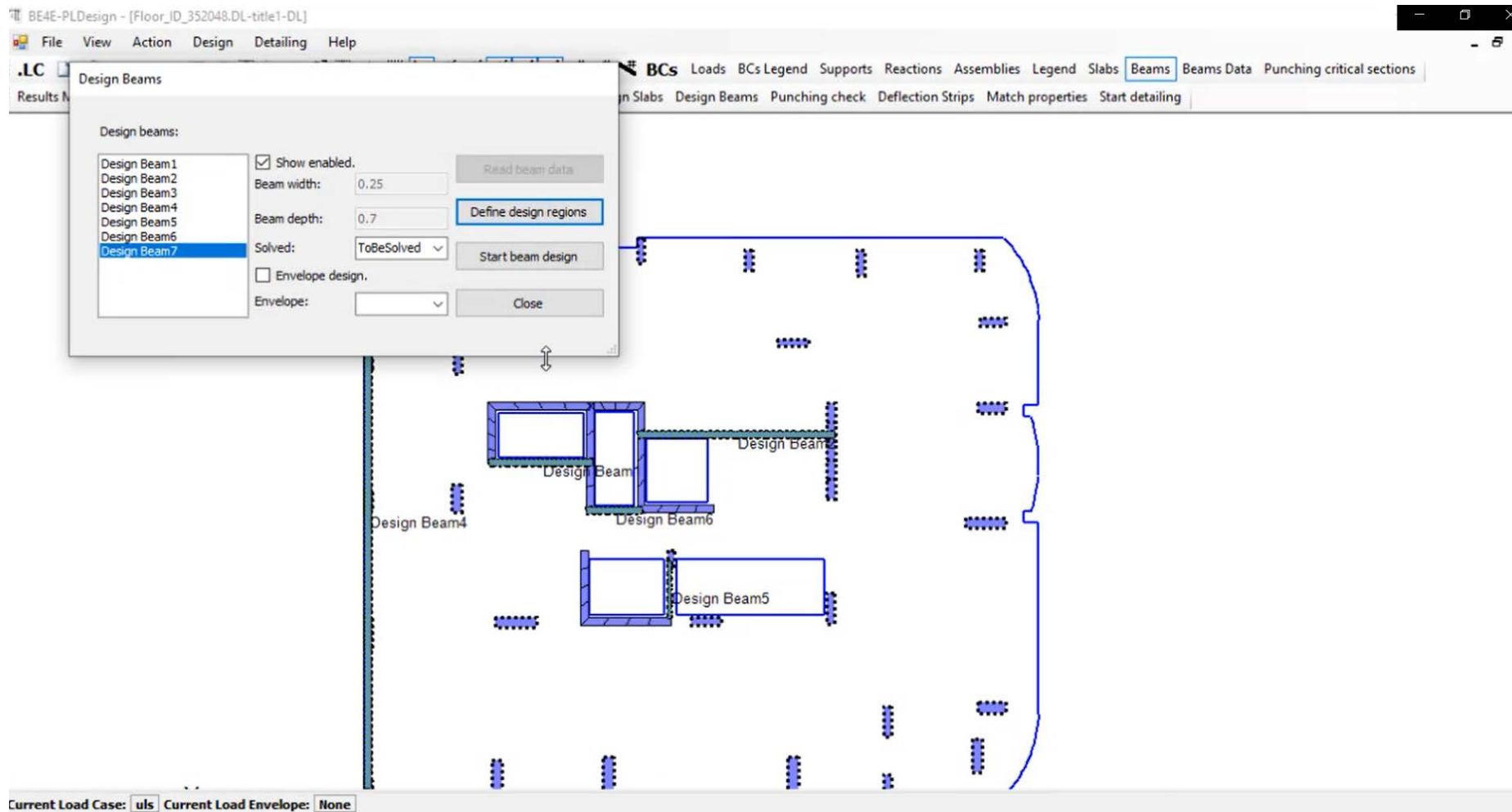


3. One floor package (BIM-PLPAK)

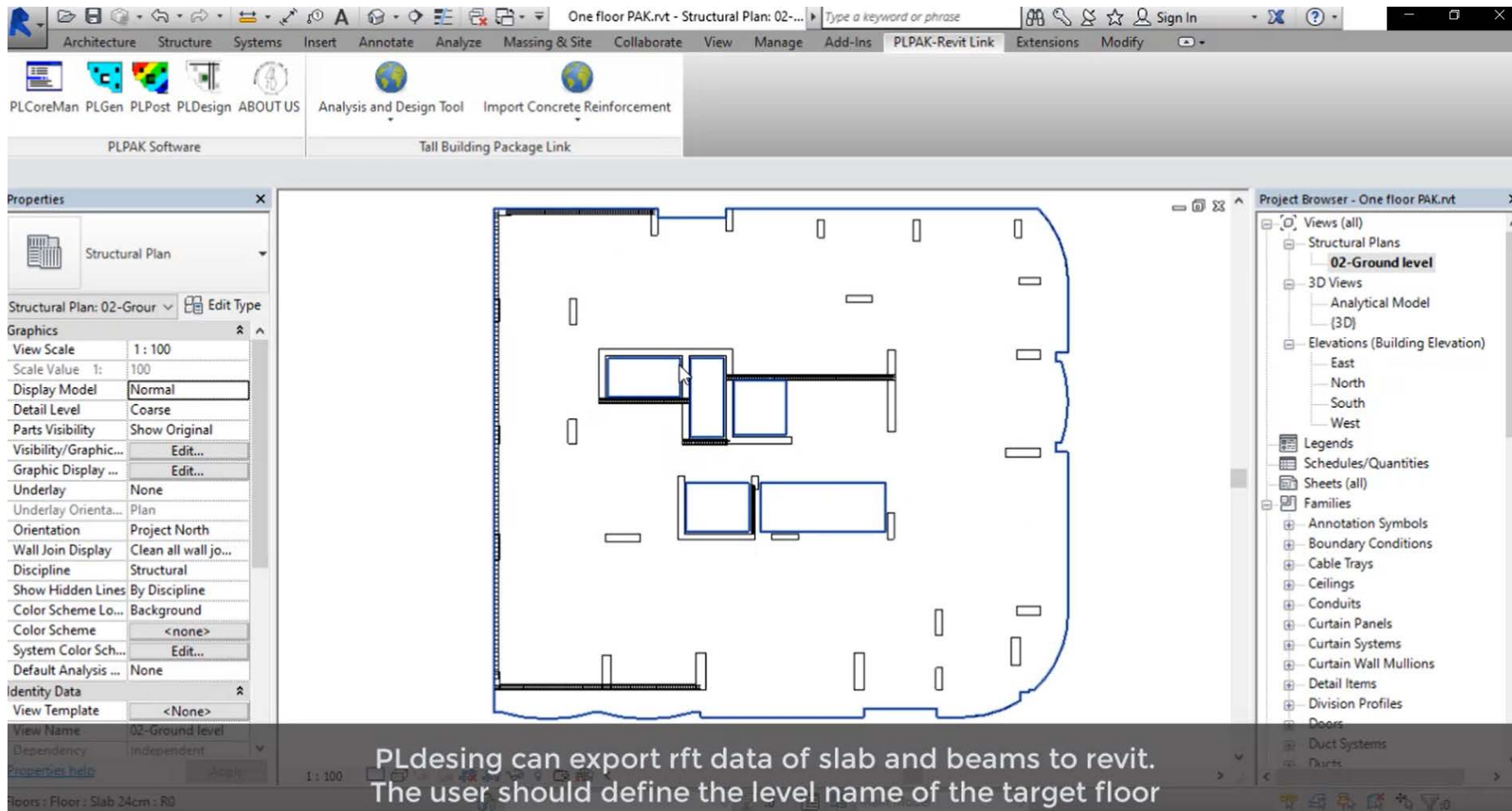


Now the software will display the required number of RFT rebar

3. One floor package (BIM-PLPAK)



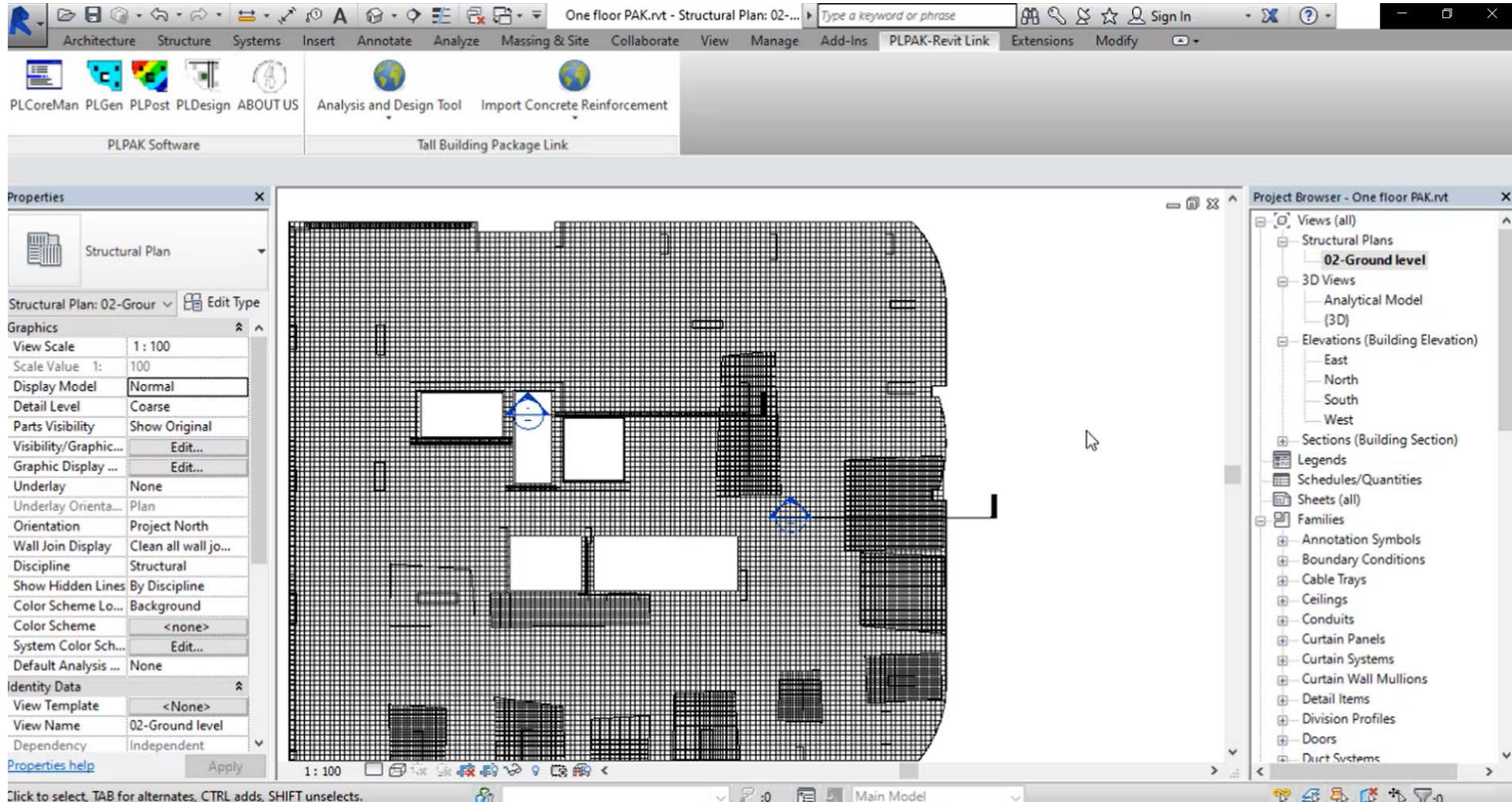
3. One floor package (BIM-PLPAK)



The screenshot displays the PLdesing software interface for a structural plan. The main window shows a blue-outlined floor plan with internal room divisions and structural elements. The interface includes a ribbon menu at the top with tabs for Architecture, Structure, Systems, Insert, Annotate, Analyze, Massing & Site, Collaborate, View, Manage, Add-Ins, PLPAK-Revit Link, Extensions, and Modify. Below the ribbon are toolbars for PLCoreMan, PLGen, PLPost, PLDesign, and ABOUT US, along with Analysis and Design Tool and Import Concrete Reinforcement. A Properties panel on the left shows settings for the Structural Plan, including View Scale (1:100), Scale Value (100), Display Model (Normal), Detail Level (Coarse), and Orientation (Project North). A Project Browser on the right lists the project hierarchy, with '02-Ground level' selected under Structural Plans. A text box at the bottom of the interface reads: "PLdesing can export rft data of slab and beams to revit. The user should define the level name of the target floor".

3. One floor package (BIM-PLPAK)

www.be4e.com



The screenshot displays the Revit software interface for a structural plan view. The ribbon at the top includes tabs for Architecture, Structure, Systems, Insert, Annotate, Analyze, Massing & Site, Collaborate, View, Manage, Add-Ins, PLPAK-Revit Link, Extensions, and Modify. The Properties panel on the left shows the 'Structural Plan' view with a scale of 1:100 and a display model of 'Normal'. The Project Browser on the right shows a tree view of the project, including '02-Ground level' and various views and families. The main view area shows a structural plan with a grid overlay and several rectangular openings representing columns and walls.

CUFE-BE



www.be4e.com

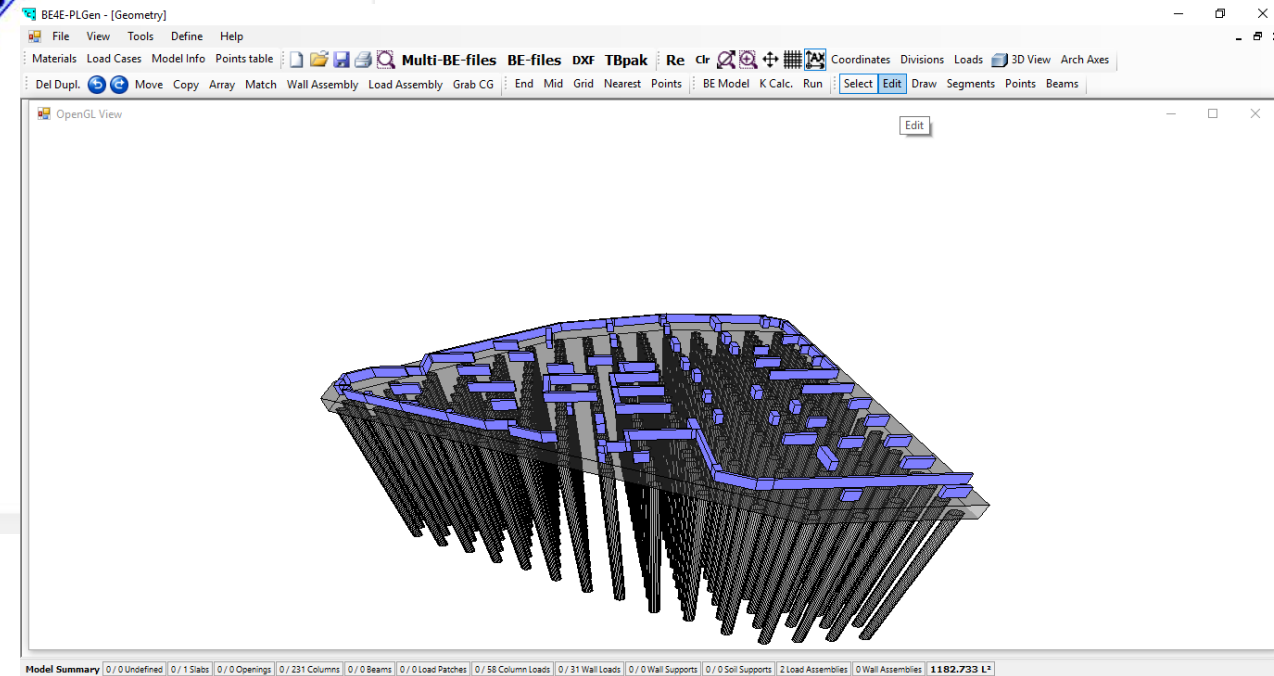
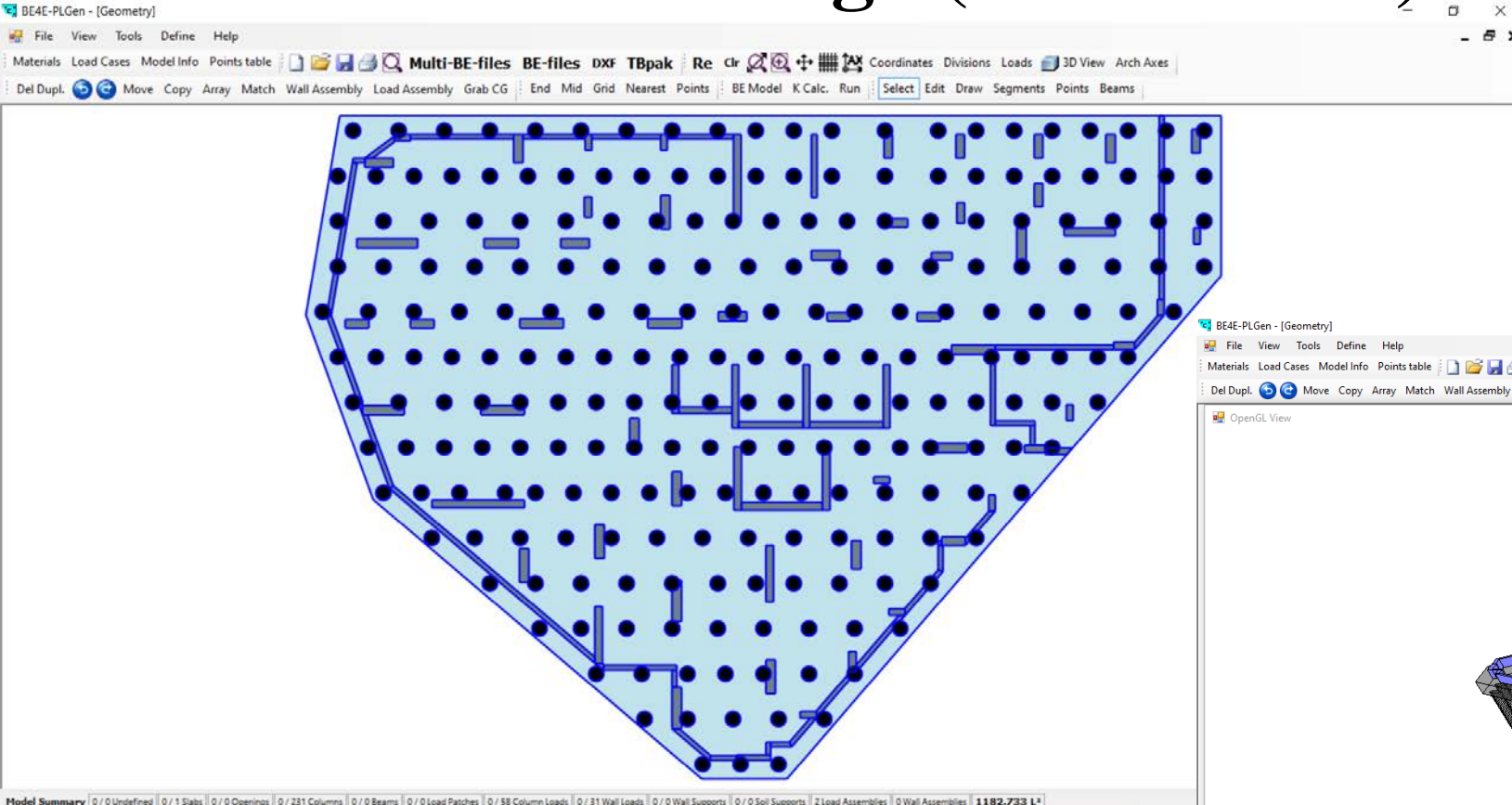


Cairo University

Table of contents

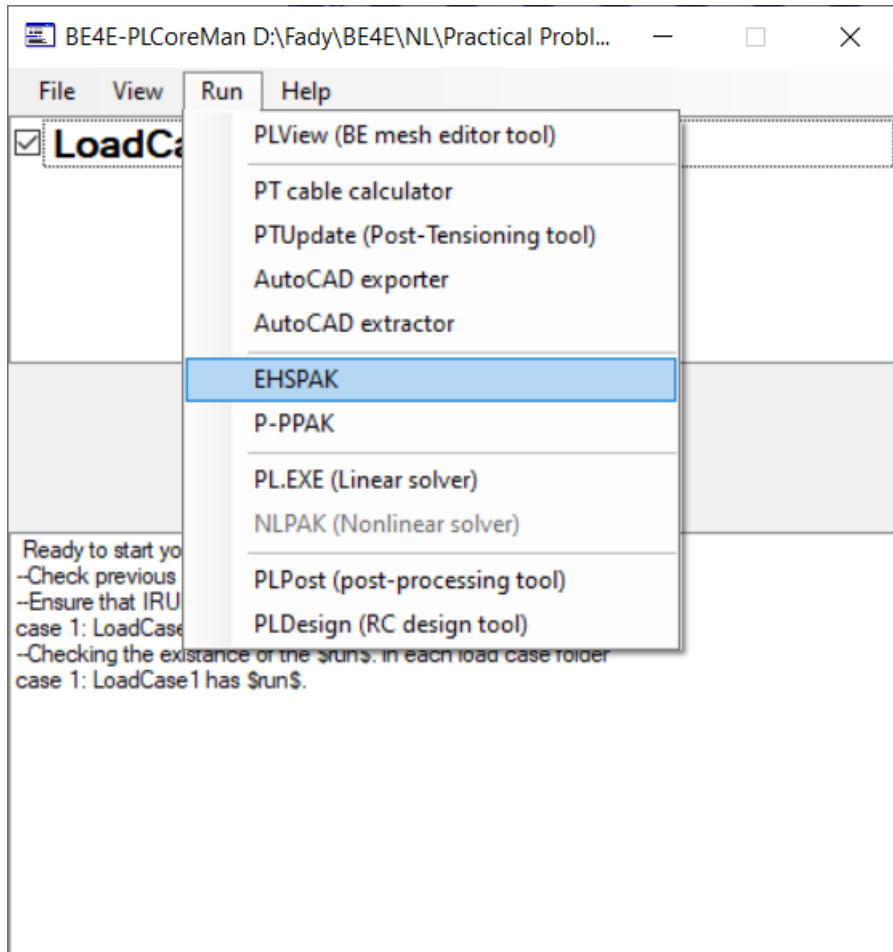
- 1. Introduction**
- 2. Basic package (PLPAK Basic)**
- 3. One floor package (BIM-PLPAK)**
- 4. Foundation Package (FoundPAK)**
5. Fixed base Package (FBPAK)
6. Post-tension Package (PTPAK)
7. Dynamic package (DynPAK)
8. Overall building package (OBPAK)
9. 4D and 5D analysis
10. Conclusions

4. Foundation Package (Foundations)

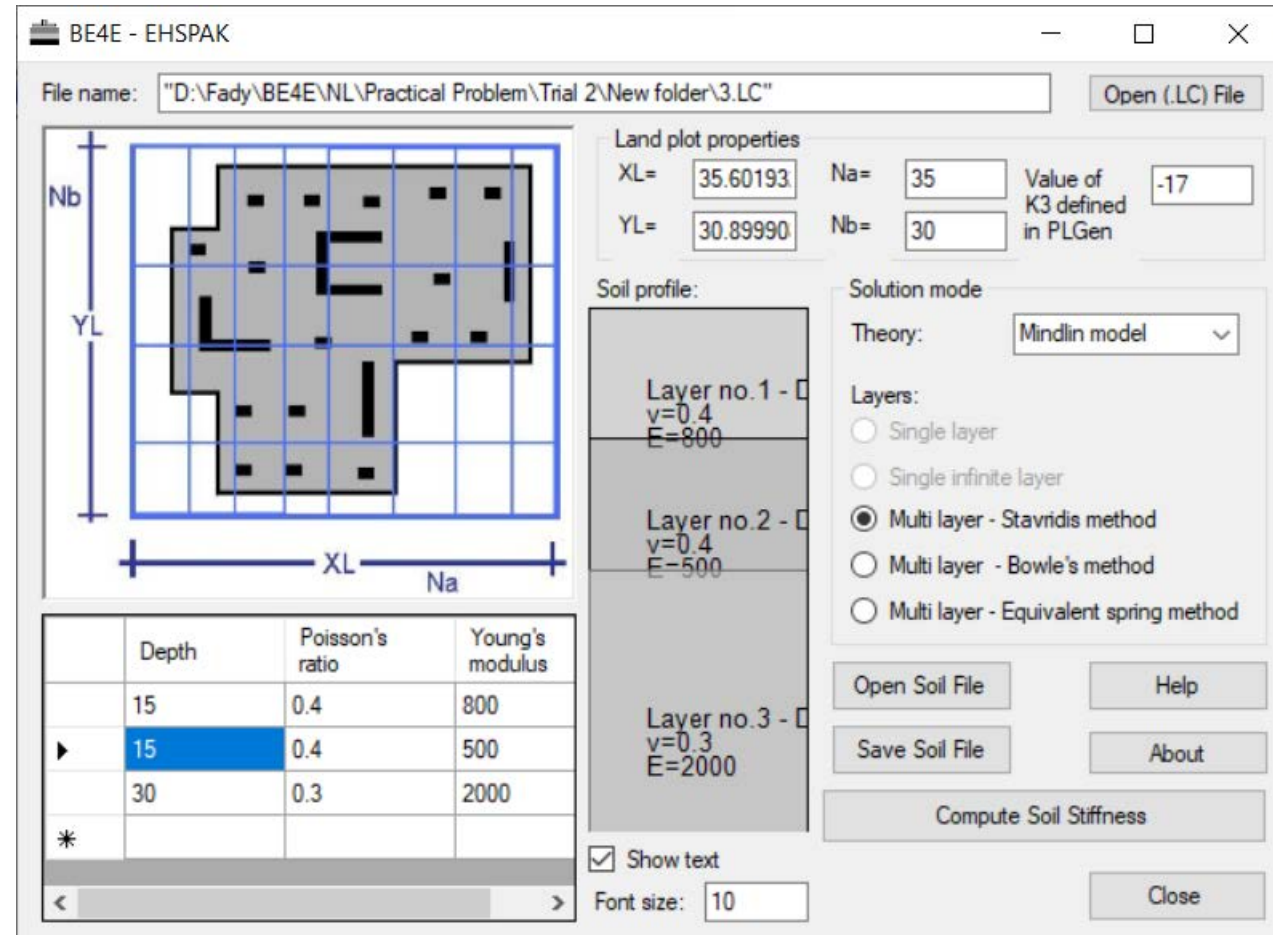


1- Generate Gen model of the piled raft defined piles as below only columns.

4. Foundation Package (Founda...)



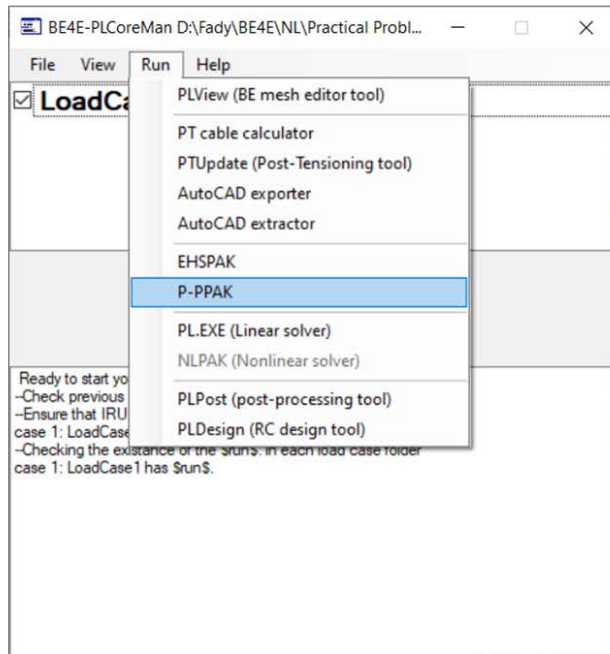
2- Run the PLCoreMan



3- Run the EHSAPK if soil exist

4. Foundation Package (Foundations)

4- Run the P-PPAK from PLCoreMan

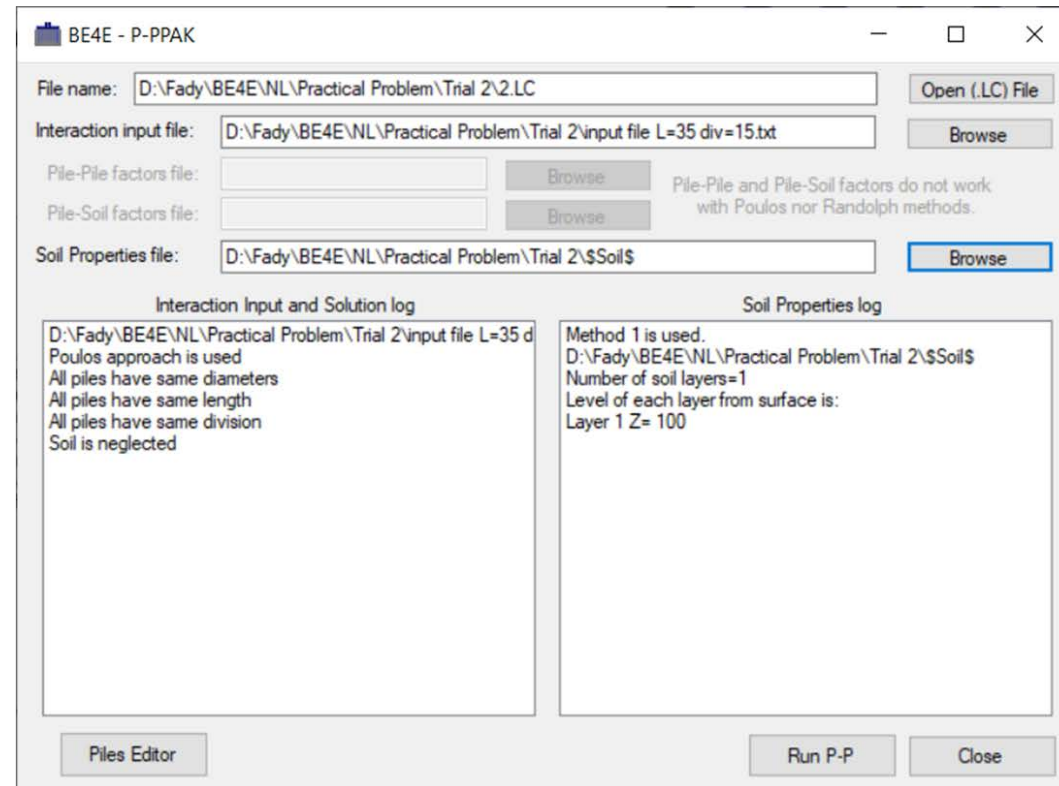


6- Load \$\$Soil\$ file that can be created by:

- Save it from EHSPAK
- User write it manually using the format presented in EHSPAK manual
- Create automatically from Revit model

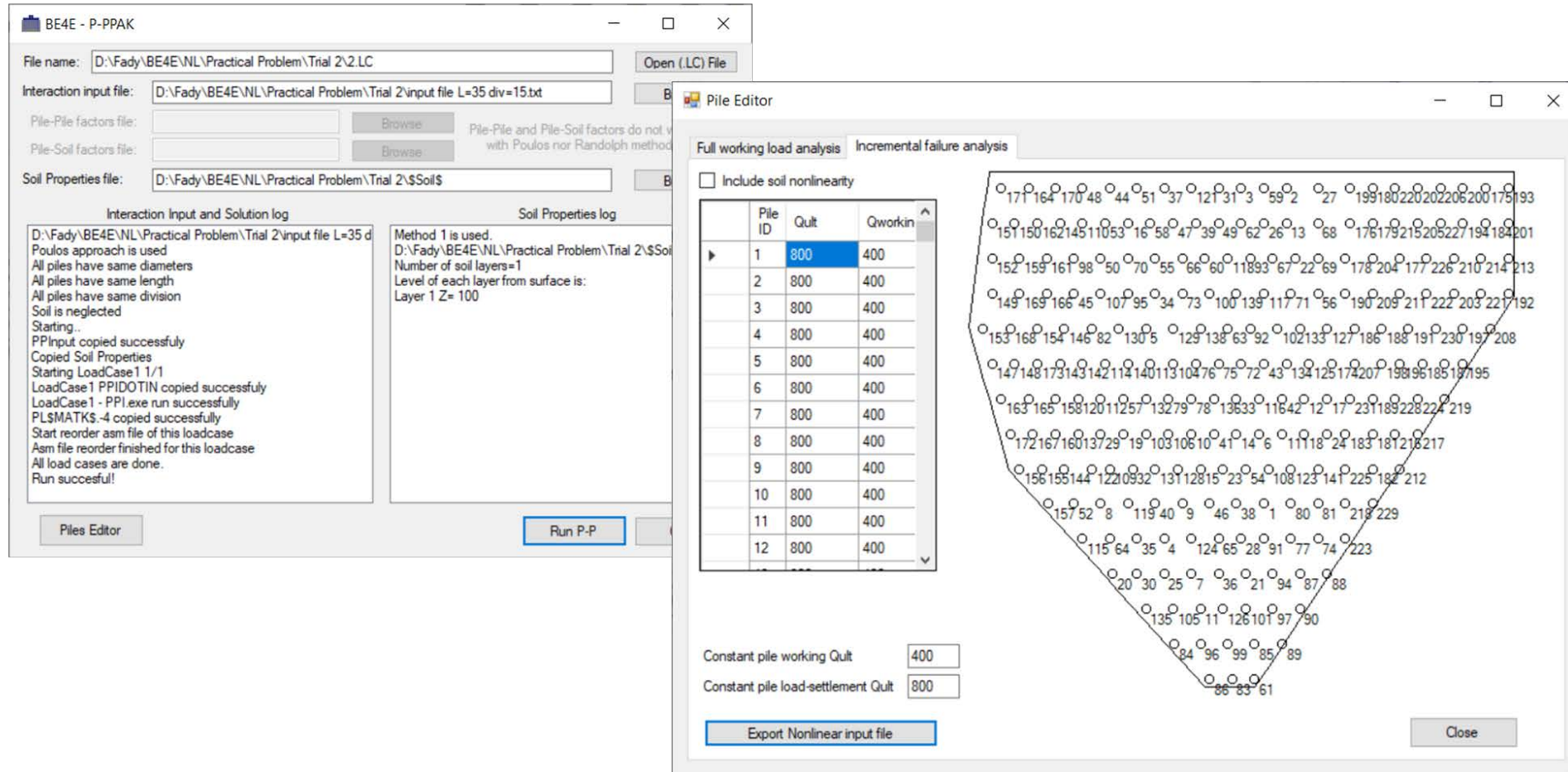
5- Load interaction effects input file that can be created by:

- User write it manually using the format presented in P-PPAK manual
- Create automatically from Revit model



4. Foundation Package (Foundations)

**8- After finishing the P-PPAK run, open the pile editor to create the NInput file
(If the nonlinear analysis will be performed)
Structure of the NInput file is presented in NLPak user manual**



The screenshot shows two windows from the BE4E software. The 'BE4E - P-PPAK' window displays the 'Interaction Input and Solution log' and 'Soil Properties log'. The log indicates that the Poulos approach is used and that the analysis was successful. The 'Soil Properties log' shows that Method 1 is used and that there is one soil layer with a depth of 100.

The 'Pile Editor' window shows a table of pile properties and a diagram of a pile group. The table lists 12 piles with a constant ultimate capacity of 800 and a constant working capacity of 400. The diagram shows a pile group with 12 piles arranged in a roughly rectangular pattern, with pile IDs ranging from 1 to 12.

Pile ID	Ult	Qworkin
1	800	400
2	800	400
3	800	400
4	800	400
5	800	400
6	800	400
7	800	400
8	800	400
9	800	400
10	800	400
11	800	400
12	800	400

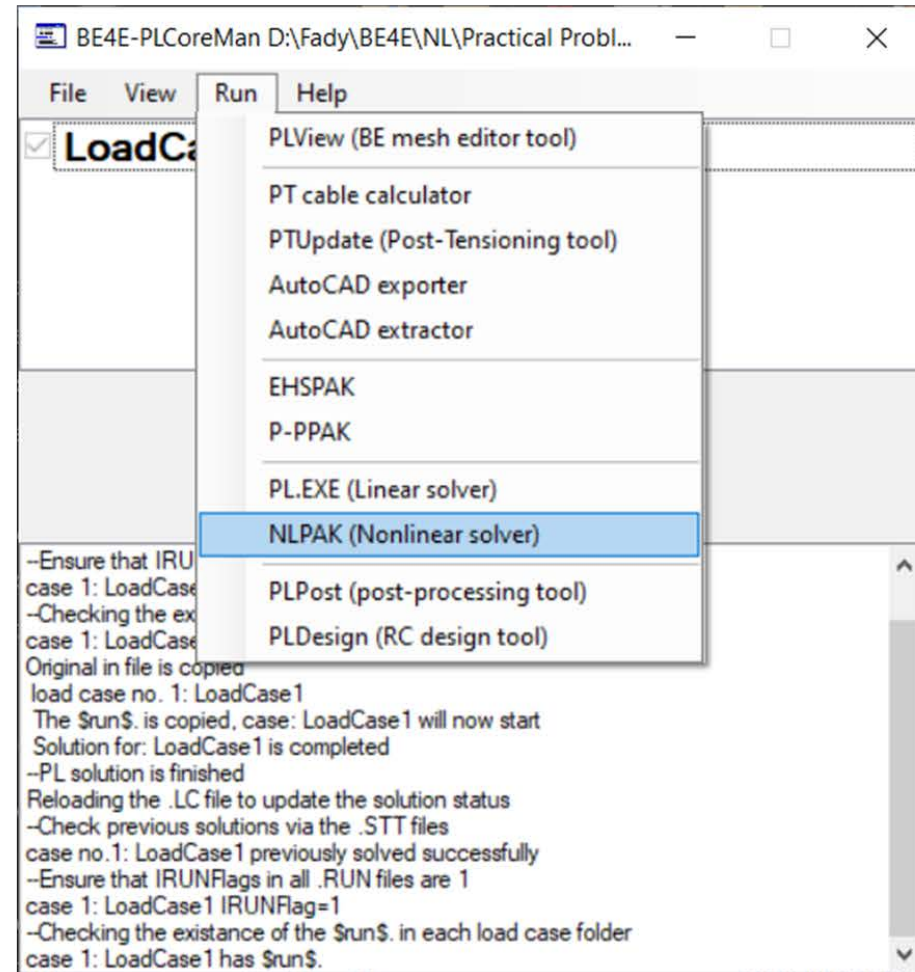
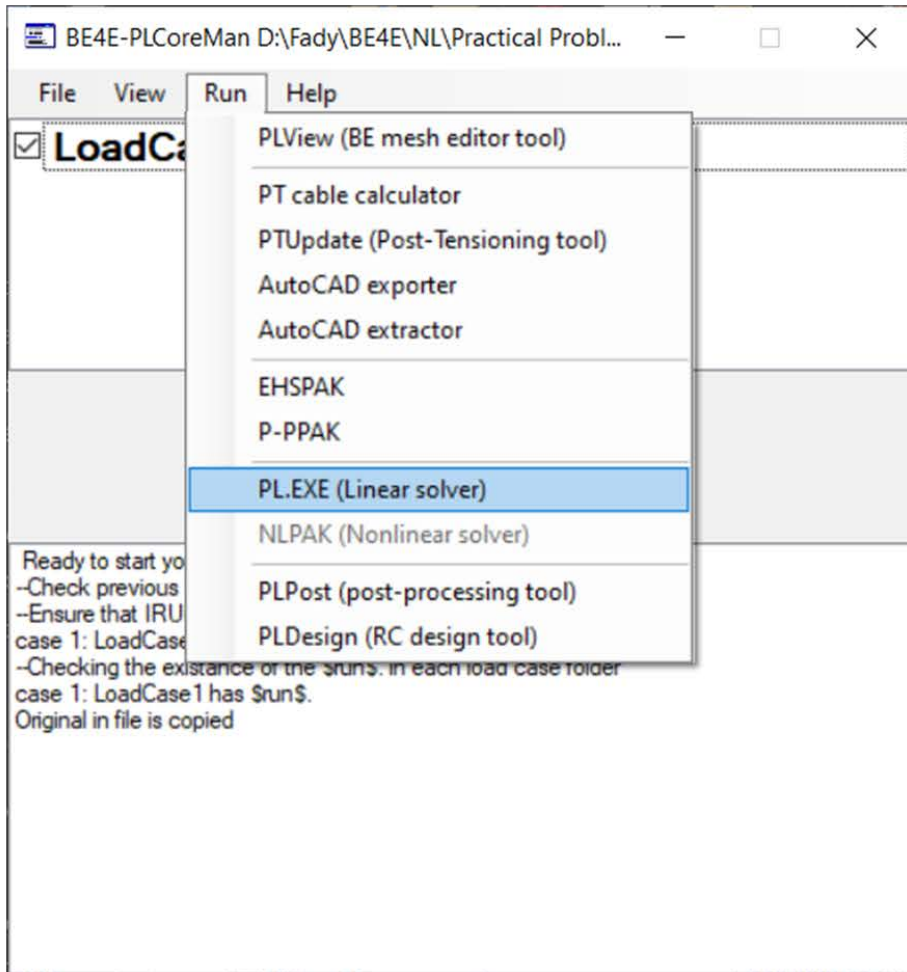
Constant pile working Qult: 400
Constant pile load-settlement Qult: 800

Buttons: Piles Editor, Run P-P, Export Nonlinear input file, Close

4. Foundation Package (Founda...)

9- Run the linear solver PL.exe

10- Run the nonlinear solver NLPK



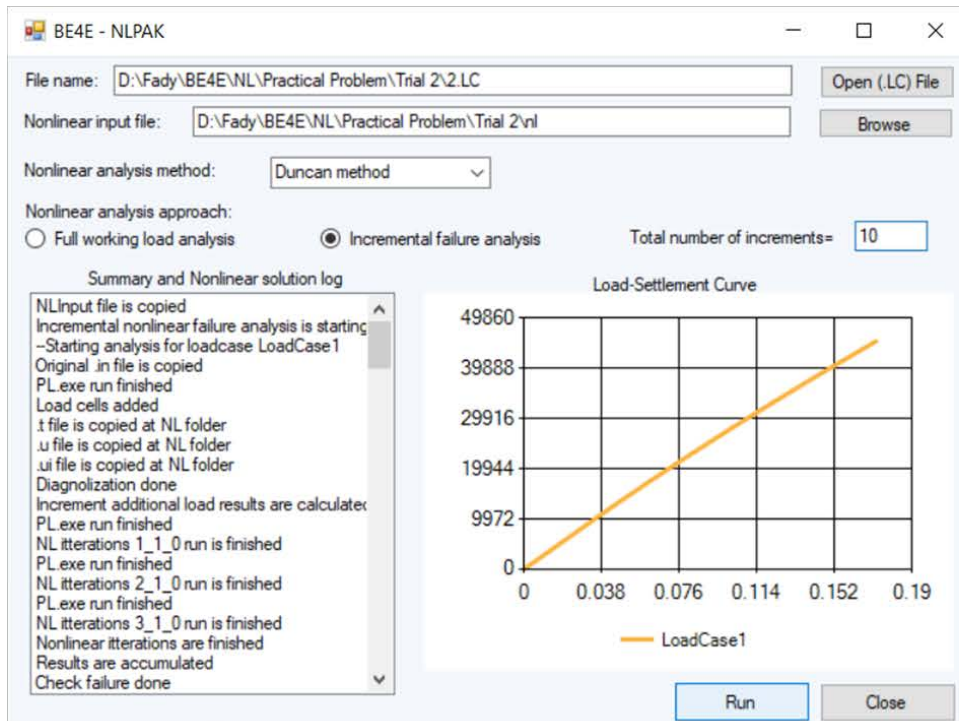
4. Foundation Package (Foundations)

11- Load the nonlinear input file

- Already created by pile editor, or
- User write it manually using the format presented in NLPak user manual

12- Choose your nonlinear approach:

- Full working load, or
- Incremental failure analysis (Input number of increments)



BE4E - NLPak

File name: D:\Fady\BE4E\NL\Practical Problem\Trial 2\2.LC

Nonlinear input file: D:\Fady\BE4E\NL\Practical Problem\Trial 2\2.LC

Nonlinear analysis method: Duncan method

Nonlinear analysis approach:
 Full working load analysis
 Incremental failure analysis
 Total number of increments= 10

Summary and Nonlinear solution log

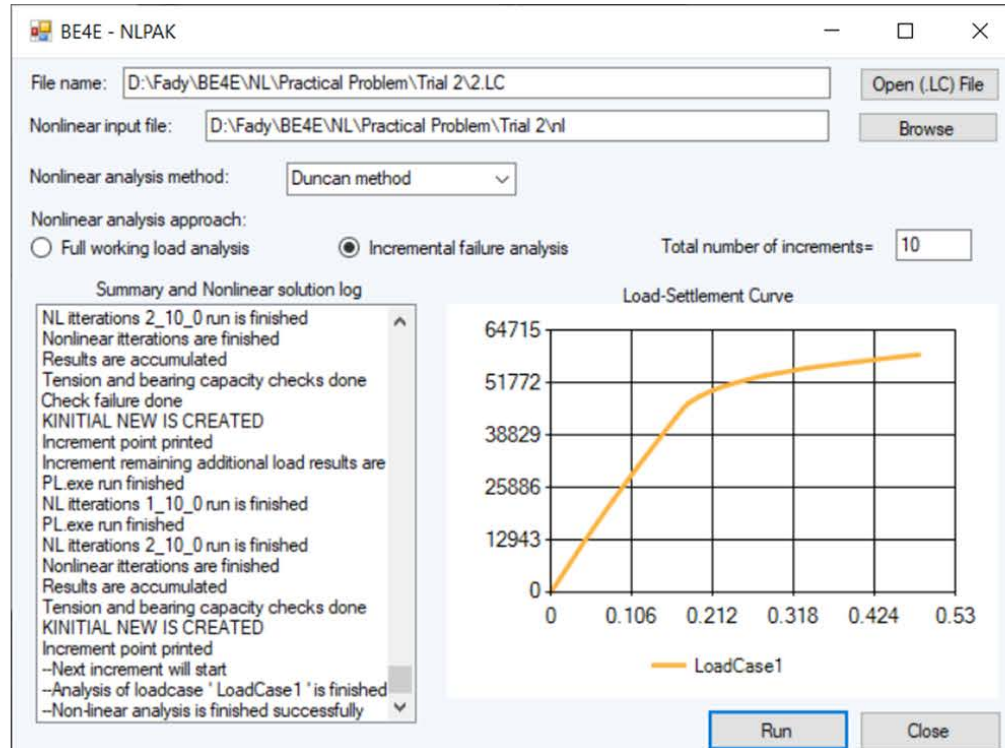
```

NLInput file is copied
Incremental nonlinear failure analysis is starting
--Starting analysis for loadcase LoadCase1
Original .in file is copied
PL.exe run finished
Load cells added
i file is copied at NL folder
u file is copied at NL folder
ui file is copied at NL folder
Diagnolization done
Increment additional load results are calculate
PL.exe run finished
NL iterations 1_1_0 run is finished
PL.exe run finished
NL iterations 2_1_0 run is finished
PL.exe run finished
NL iterations 3_1_0 run is finished
Nonlinear iterations are finished
Results are accumulated
Check failure done
    
```

Load-Settlement Curve

Settlement (m)	Load (kN)
0	0
0.038	9972
0.076	19944
0.114	29916
0.152	39888
0.19	49860

— LoadCase1



BE4E - NLPak

File name: D:\Fady\BE4E\NL\Practical Problem\Trial 2\2.LC

Nonlinear input file: D:\Fady\BE4E\NL\Practical Problem\Trial 2\2.LC

Nonlinear analysis method: Duncan method

Nonlinear analysis approach:
 Full working load analysis
 Incremental failure analysis
 Total number of increments= 10

Summary and Nonlinear solution log

```

NL iterations 2_10_0 run is finished
Nonlinear iterations are finished
Results are accumulated
Tension and bearing capacity checks done
Check failure done
KINITIAL NEW IS CREATED
Increment point printed
Increment remaining additional load results are
PL.exe run finished
NL iterations 1_10_0 run is finished
PL.exe run finished
NL iterations 2_10_0 run is finished
Nonlinear iterations are finished
Results are accumulated
Tension and bearing capacity checks done
KINITIAL NEW IS CREATED
Increment point printed
--Next increment will start
--Analysis of loadcase ' LoadCase1 ' is finished
--Non-linear analysis is finished successfully
    
```

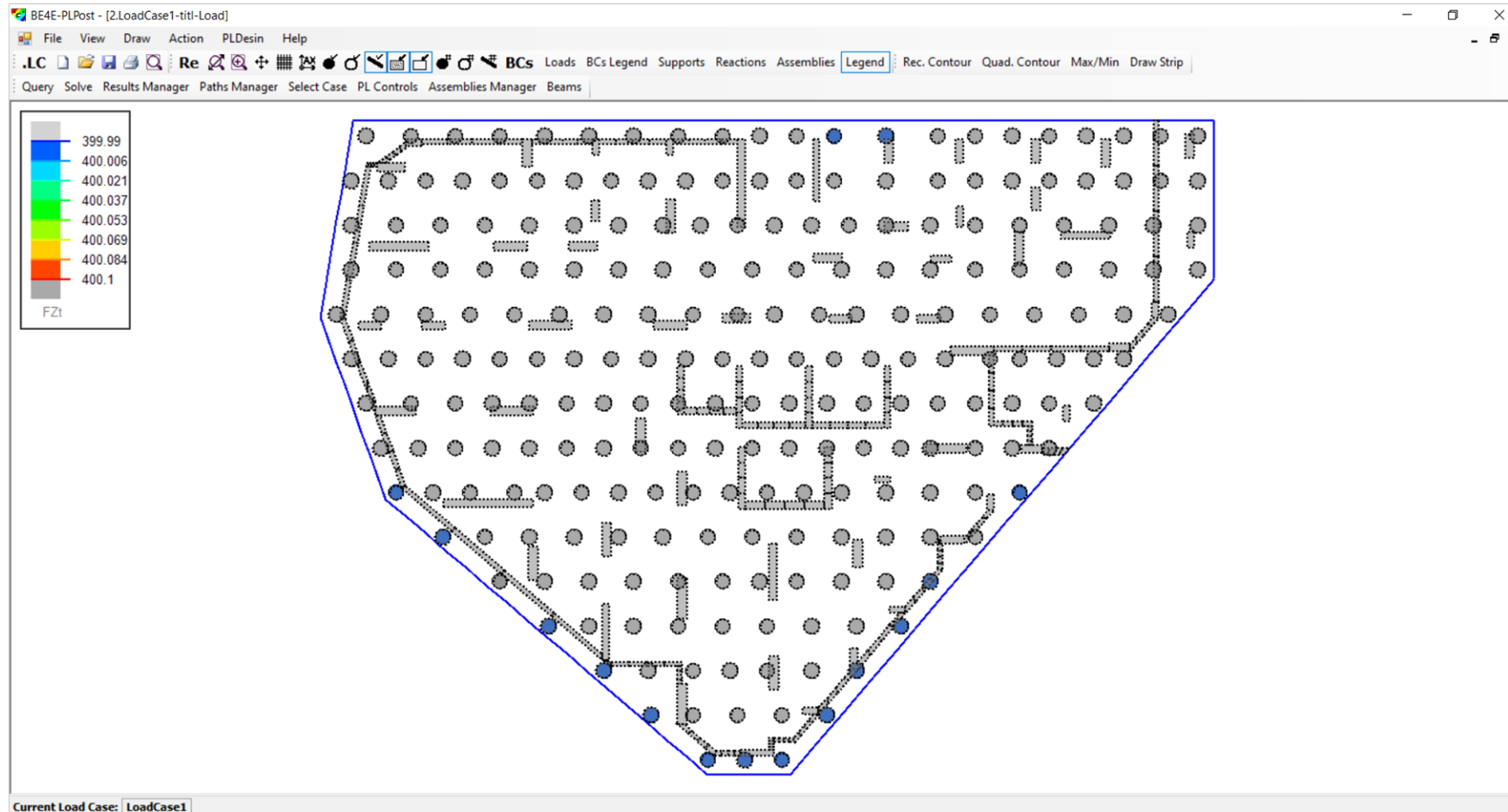
Load-Settlement Curve

Settlement (m)	Load (kN)
0	0
0.106	12943
0.212	25886
0.318	38829
0.424	51772
0.53	64715

— LoadCase1

4. Foundation Package (Foundation)

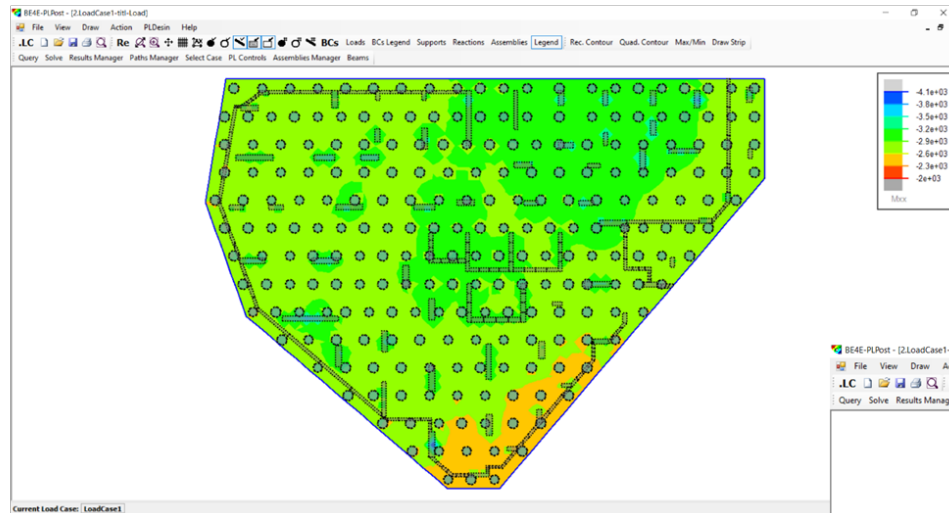
13- Run PLPost to see your results



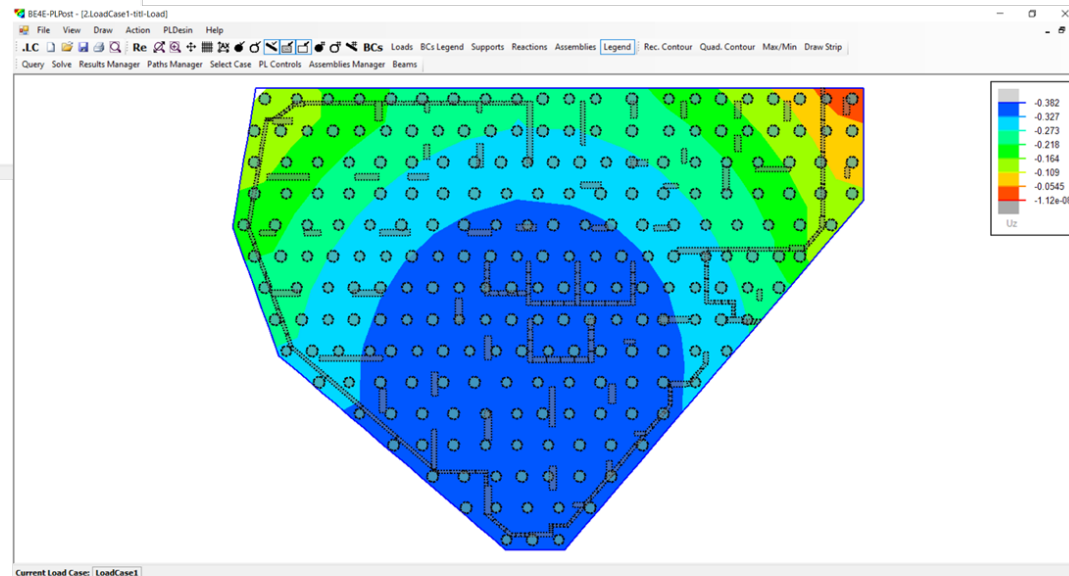
Piles failed in blue color

4. Foundation Package (FoundationK)

13- Run PLPost to see your results



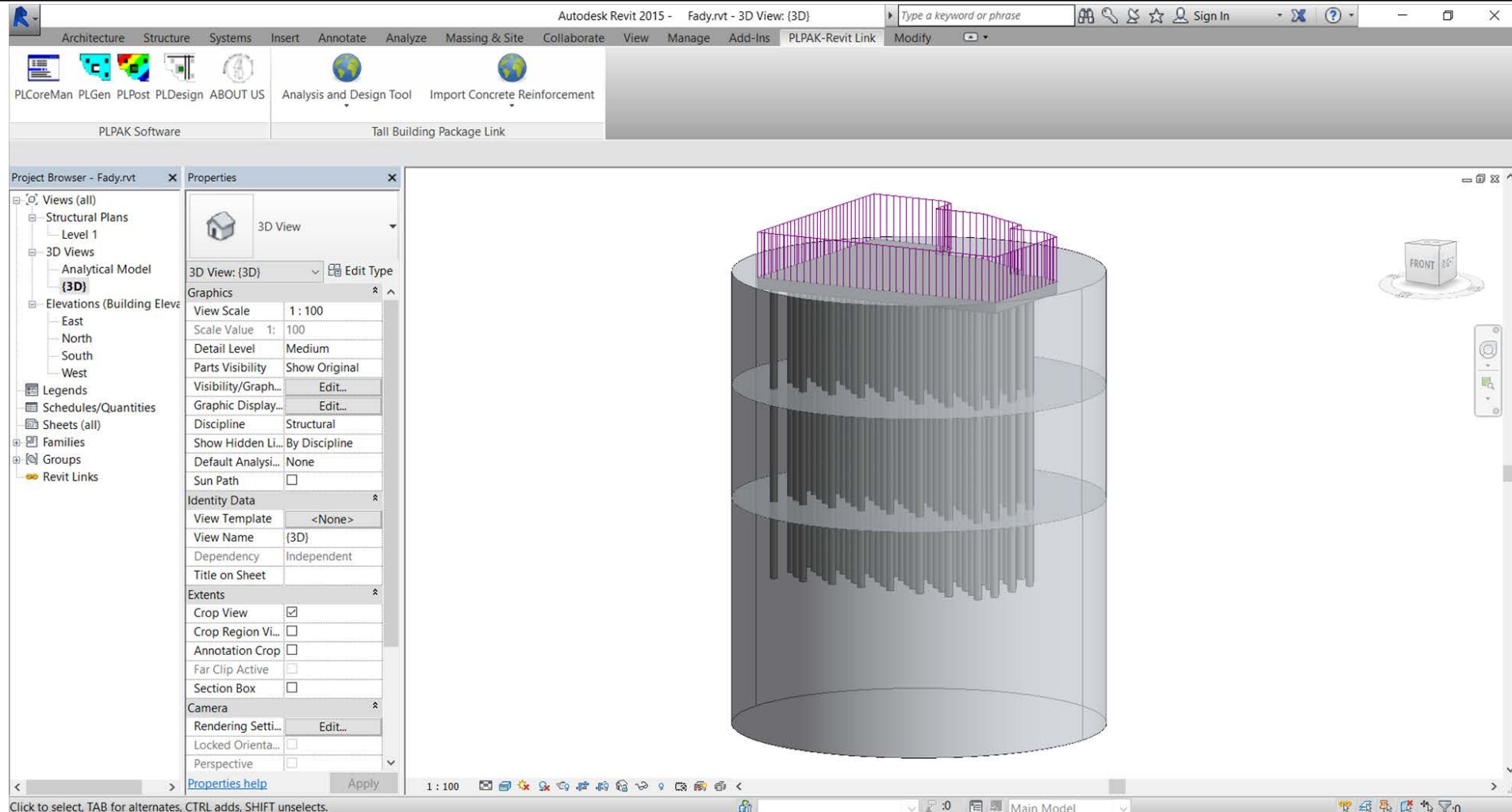
Nonlinear bending moment M_{xx} contours



Nonlinear settlement contours

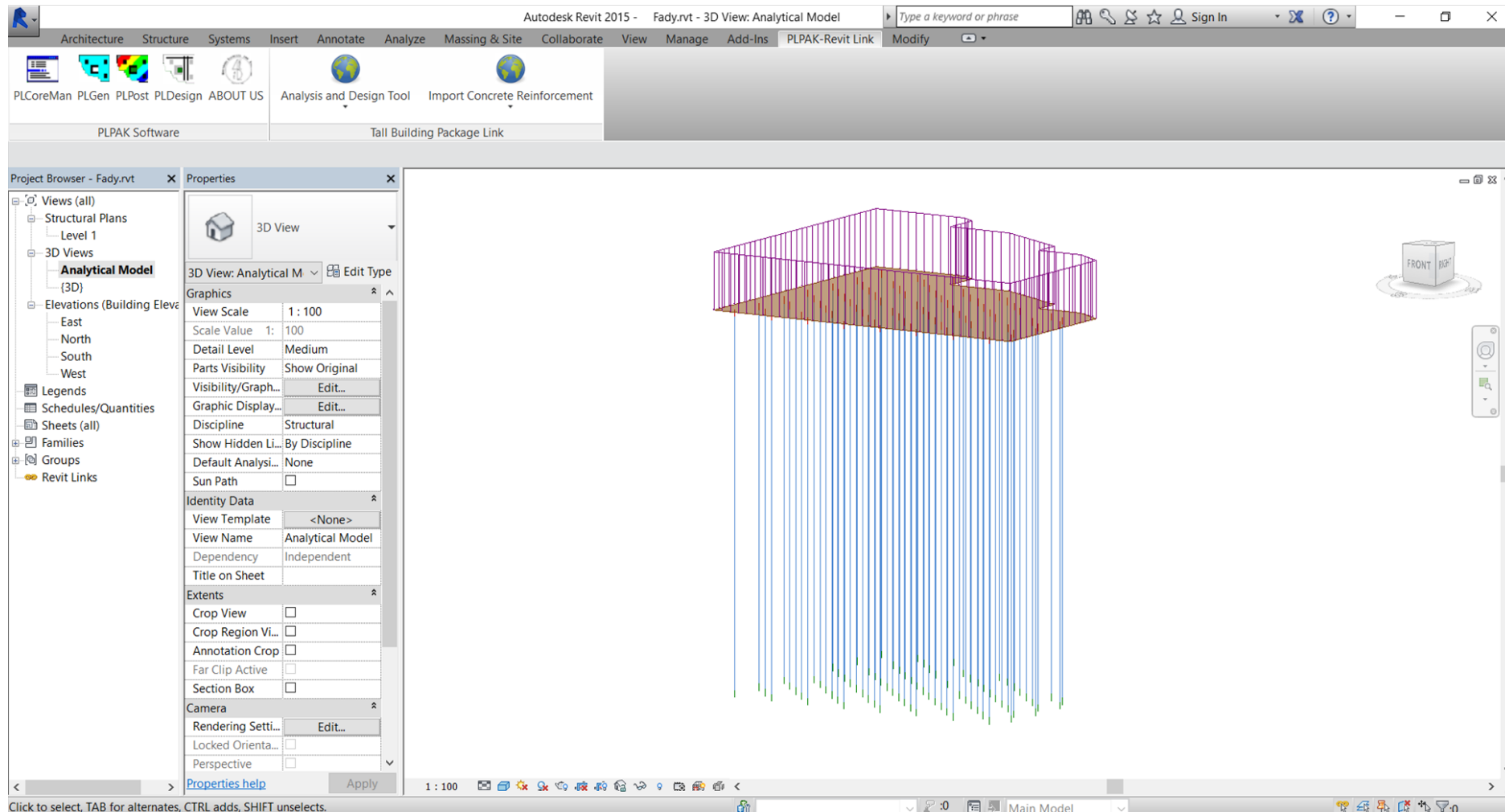
4. Foundation Package (Foundation)

1- Generate piled raft model using Revit



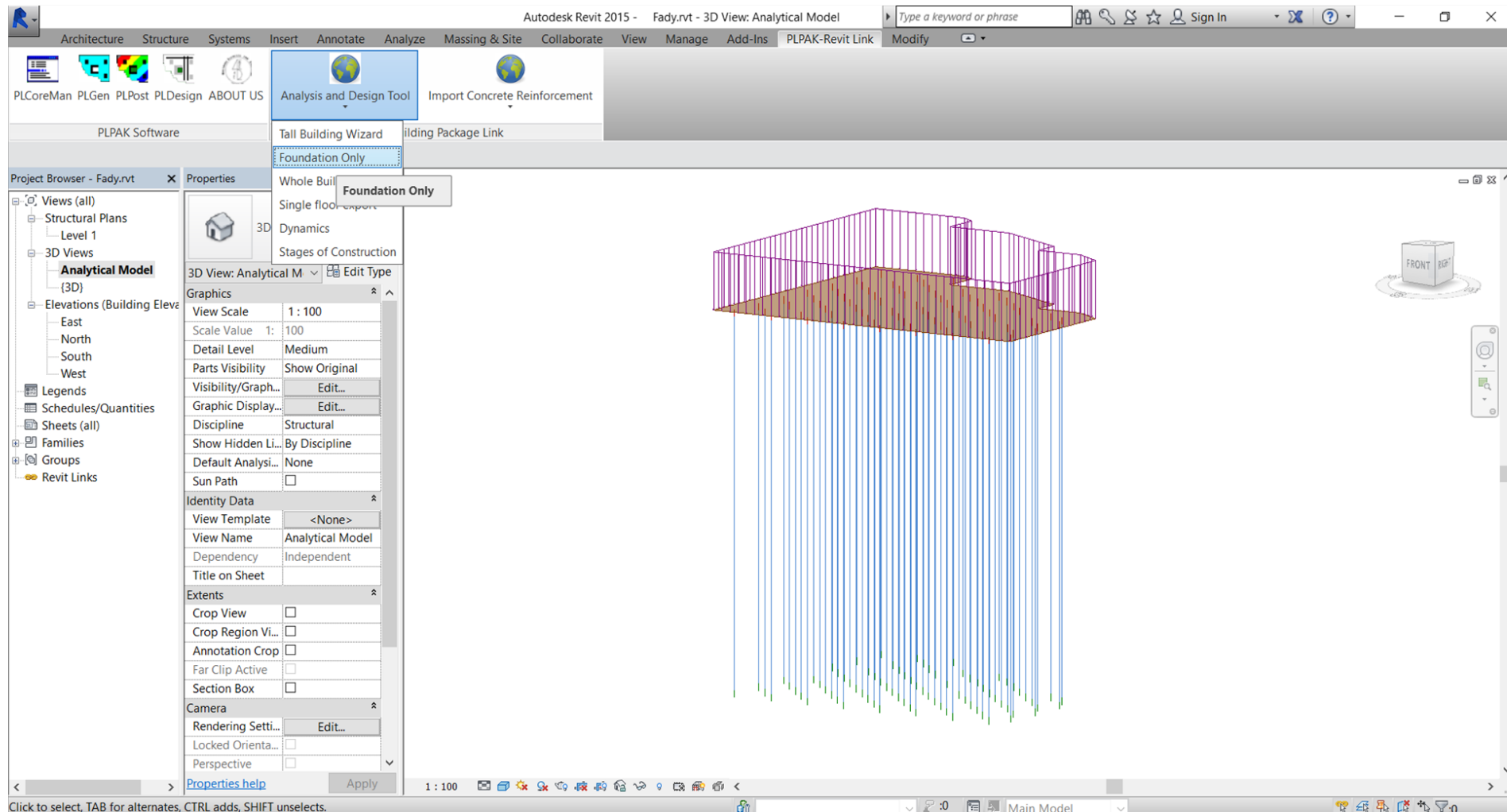
4. Foundation Package (Foundation)

Revit analytical model



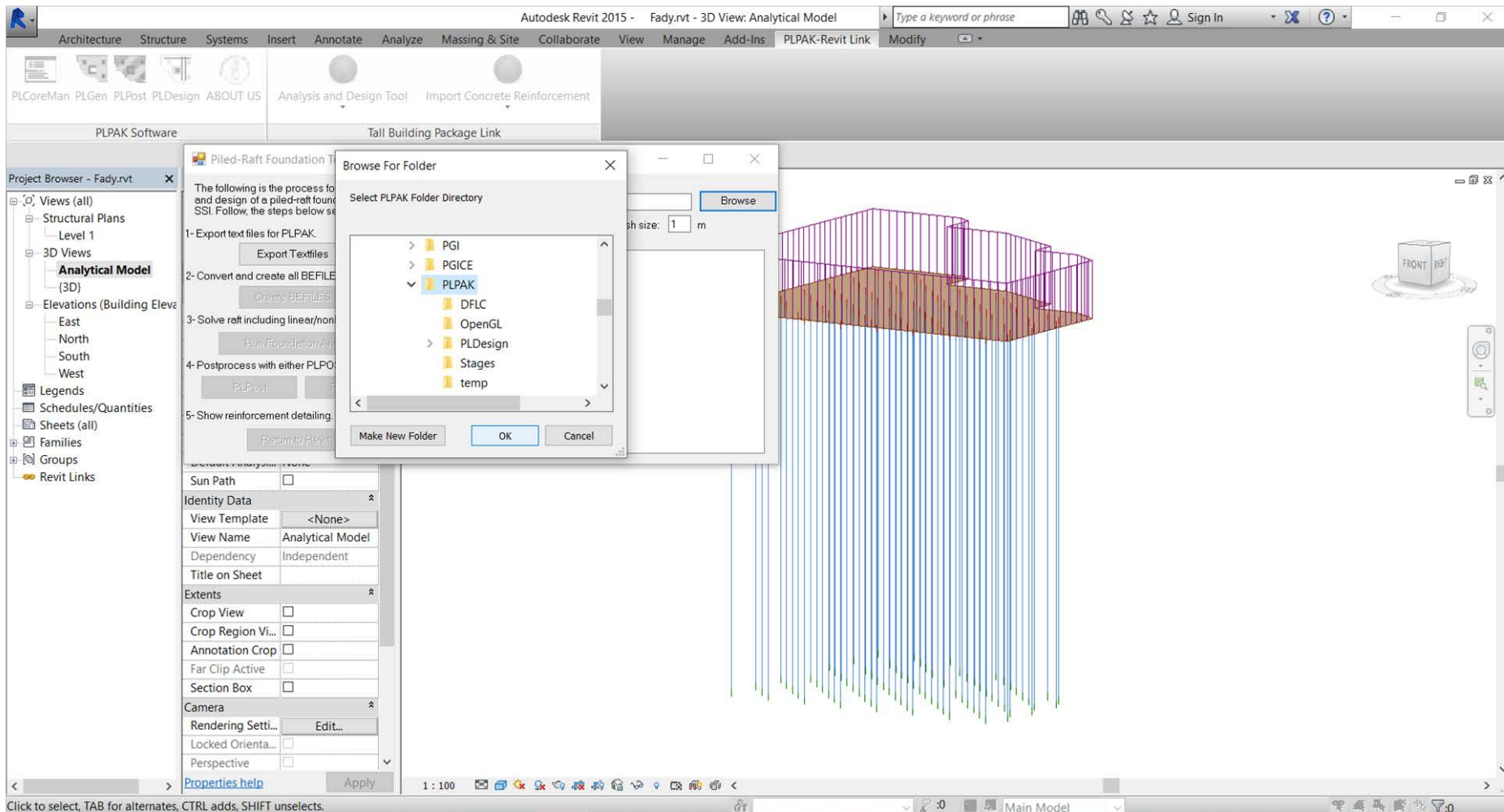
4. Foundation Package (Foundation Framework)

2- Open foundation only wizard from analysis and design PLPAK tool add-ins



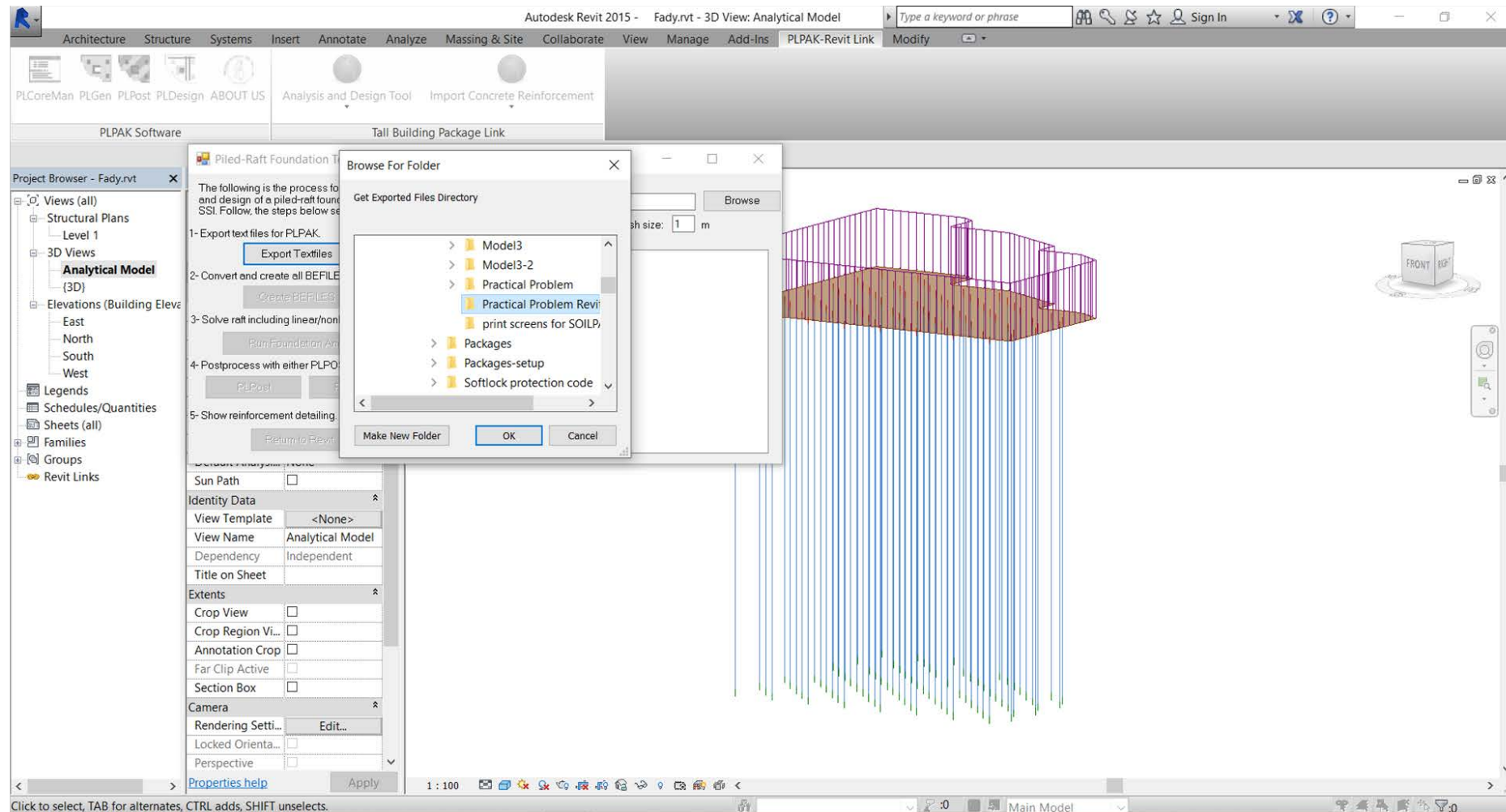
4. Foundation Package (Foundations)

3- Browse the PLPAK folder path (if not the default C:\program files\PLPAK)



4. Foundation Package (Foundation)

4- Export text files, and browse the problem folder (after defined the piles and soil divisions)



The screenshot displays the Autodesk Revit 2015 interface with the Piled-Raft Foundation tool active. The tool's process is outlined as follows:

- 1- Export text files for PLPAK. (Export Textfiles button is highlighted)
- 2- Convert and create all BEFILES. (Create BEFILES button)
- 3- Solve raft including linear/non-linear analysis. (Run Foundation Analysis button)
- 4- Postprocess with either PLPost or PLDesign. (PLPost button)
- 5- Show reinforcement detailing. (Return to Revit button)

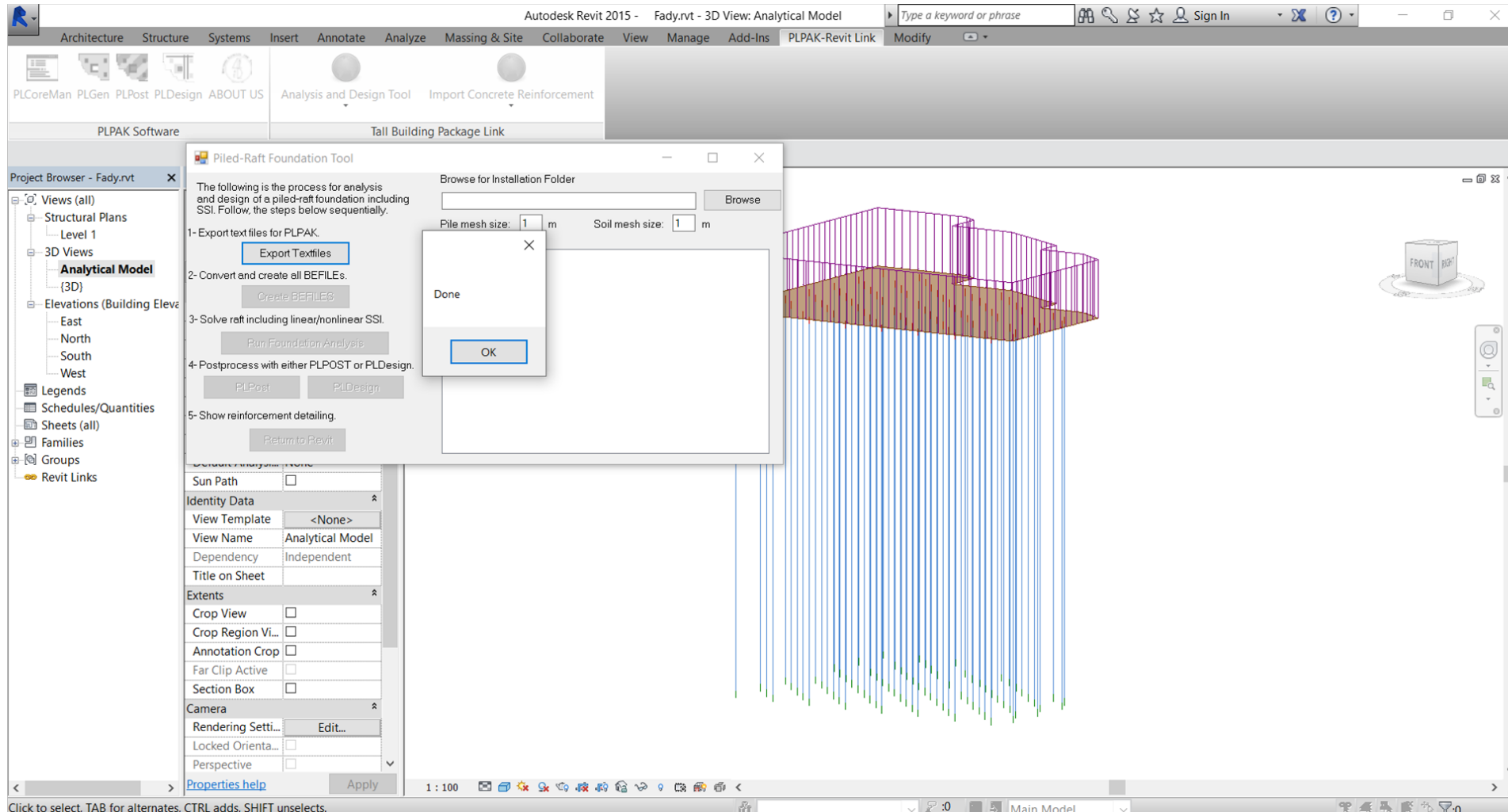
The 'Browse For Folder' dialog box is open, showing the following file tree structure:

- Model3
- Model3-2
- Practical Problem
- Practical Problem Revit (selected)
- print screens for SOILP
- Packages
- Packages-setup
- Softlock protection code

The background shows a 3D analytical model of a foundation with piles and soil divisions. The interface includes a Project Browser on the left, a Properties panel at the bottom, and a ribbon at the top with tabs for Architecture, Structure, Systems, Insert, Annotate, Analyze, Massing & Site, Collaborate, View, Manage, Add-Ins, and PLPAK-Revit Link.

4. Foundation Package (Foundations) www.be4e.com

Exporting text files done



Autodesk Revit 2015 - Fady.rvt - 3D View: Analytical Model

Architecture Structure Systems Insert Annotate Analyze Massing & Site Collaborate View Manage Add-Ins PLPAK-Revit Link Modify

PLCoreMan PLGen PLPost PLDesign ABOUT US Analysis and Design Tool Import Concrete Reinforcement

PLPAK Software Tall Building Package Link

Project Browser - Fady.rvt

- Views (all)
- Structural Plans
 - Level 1
- 3D Views
 - Analytical Model (3D)**
- Elevations (Building Elev)
 - East
 - North
 - South
 - West
- Legends
- Schedules/Quantities
- Sheets (all)
- Families
- Groups
- Revit Links

Piled-Raft Foundation Tool

The following is the process for analysis and design of a piled-raft foundation including SSL. Follow the steps below sequentially.

- 1- Export text files for PLPAK.
Export Textfiles
- 2- Convert and create all BEFILES.
Create BEFILES
- 3- Solve raft including linear/nonlinear SSL.
Run Foundation Analysis
- 4- Postprocess with either PLPOST or PLDesign.
PLPost PLDesign
- 5- Show reinforcement detailing.
Return to Revit

Browse for Installation Folder

File mesh size: 1 m Soil mesh size: 1 m

Done

OK

Sun Path

Identity Data

View Template	<None>
View Name	Analytical Model
Dependency	Independent
Title on Sheet	

Extents

Crop View	<input type="checkbox"/>
Crop Region Vi...	<input type="checkbox"/>
Annotation Crop	<input type="checkbox"/>
Far Clip Active	<input type="checkbox"/>
Section Box	<input type="checkbox"/>

Camera

Rendering Setti...	Edit...
Locked Orienta...	<input type="checkbox"/>
Perspective	<input type="checkbox"/>

Properties help Apply

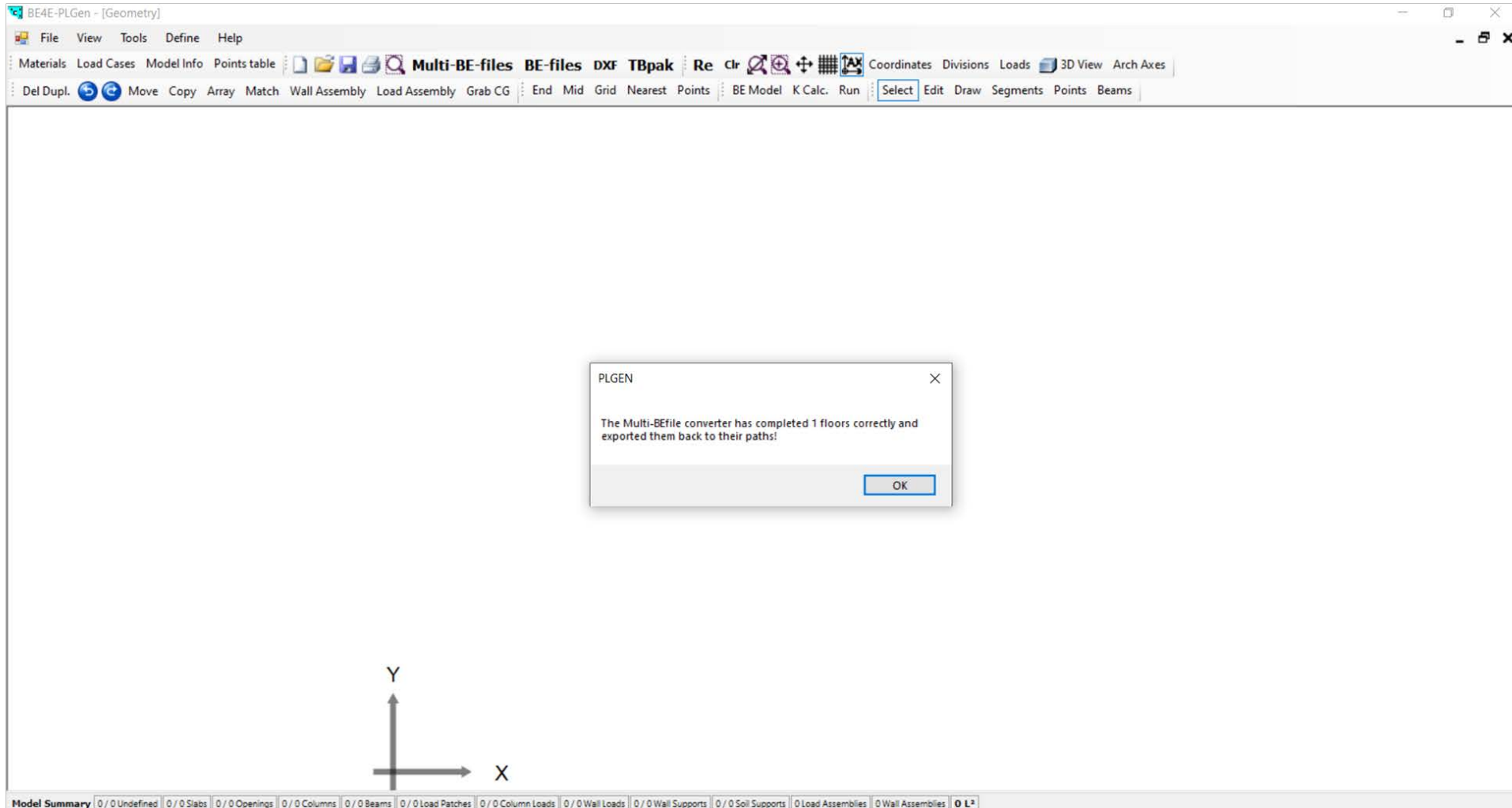
1 : 100

Click to select, TAB for alternates, CTRL adds, SHIFT unselects.

Main Model

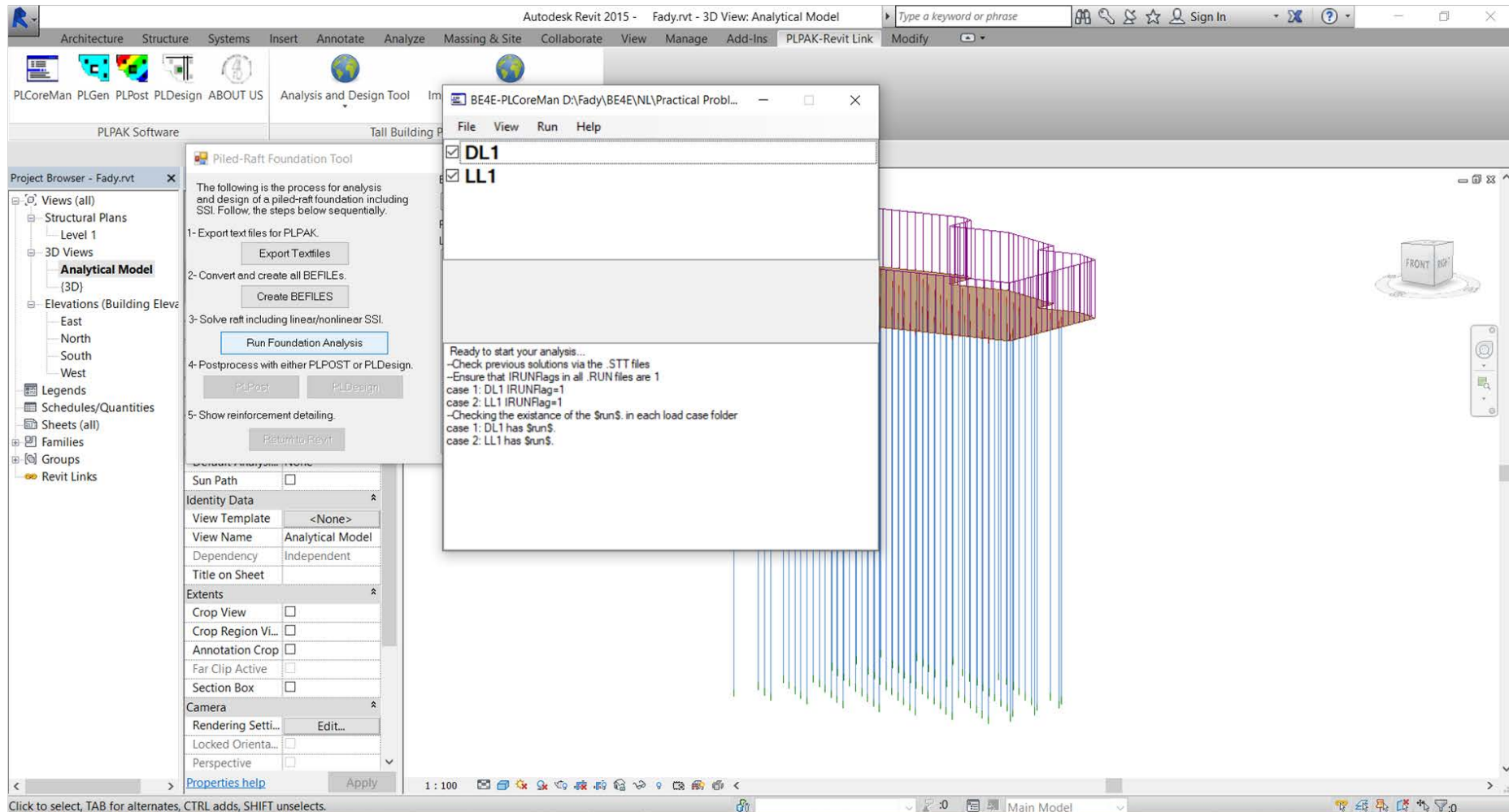
4. Foundation Package (Foundation)

5- Create BEFiles



4. Foundation Package (FounderaK)

6- Run foundation analysis (open the PLCoreMan automatically)



The screenshot displays the Autodesk Revit 2015 interface with the PLCoreMan software running. The main window shows a 3D analytical model of a building foundation with a grid of piles extending downwards. A 'Piled-Raft Foundation Tool' dialog box is open, providing instructions for the analysis process:

- 1- Export text files for PLPAK. (Export Textfiles button)
- 2- Convert and create all BEFILES. (Create BEFILES button)
- 3- Solve raft including linear/nonlinear SSI. (Run Foundation Analysis button)
- 4- Postprocess with either PLPOST or PLDesign. (PLPost, PLDesign buttons)
- 5- Show reinforcement detailing. (Return to Revit button)

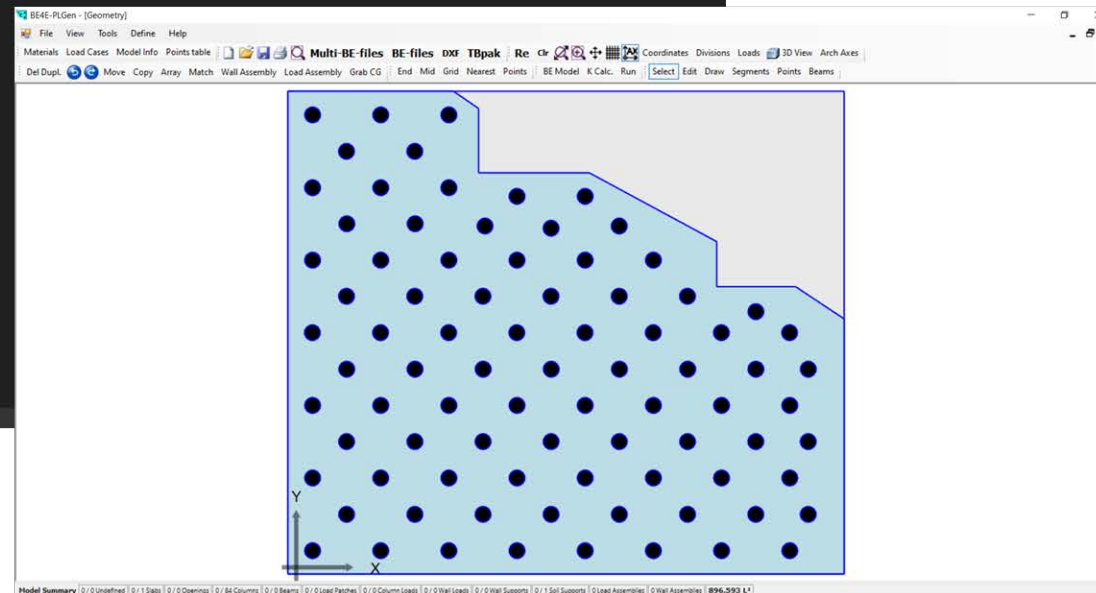
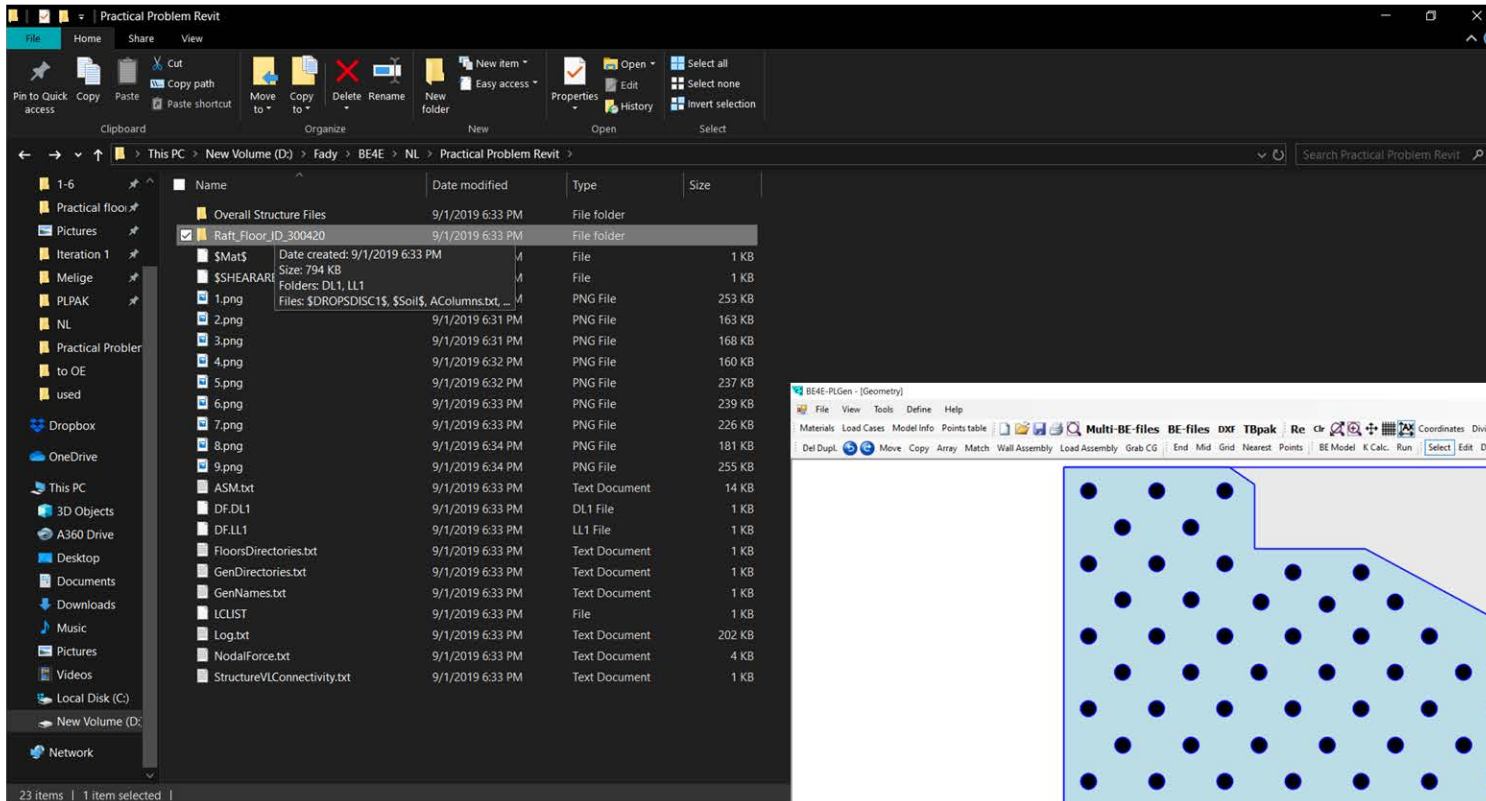
Below the instructions are various settings for the analysis, including Sun Path, Identity Data, Extents, and Camera options. A 'Ready to start your analysis...' message is displayed, listing the following instructions:

- Check previous solutions via the .STT files
- Ensure that IRUNFlag=1 in all .RUN files are 1
- case 1: DL1 IRUNFlag=1
- case 2: LL1 IRUNFlag=1
- Checking the existence of the \$run\$. in each load case folder
- case 1: DL1 has \$run\$.
- case 2: LL1 has \$run\$.

The background shows the Revit interface with the 'Analytical Model' view selected in the Project Browser. The status bar at the bottom indicates 'Main Model' and '1: 100' scale.

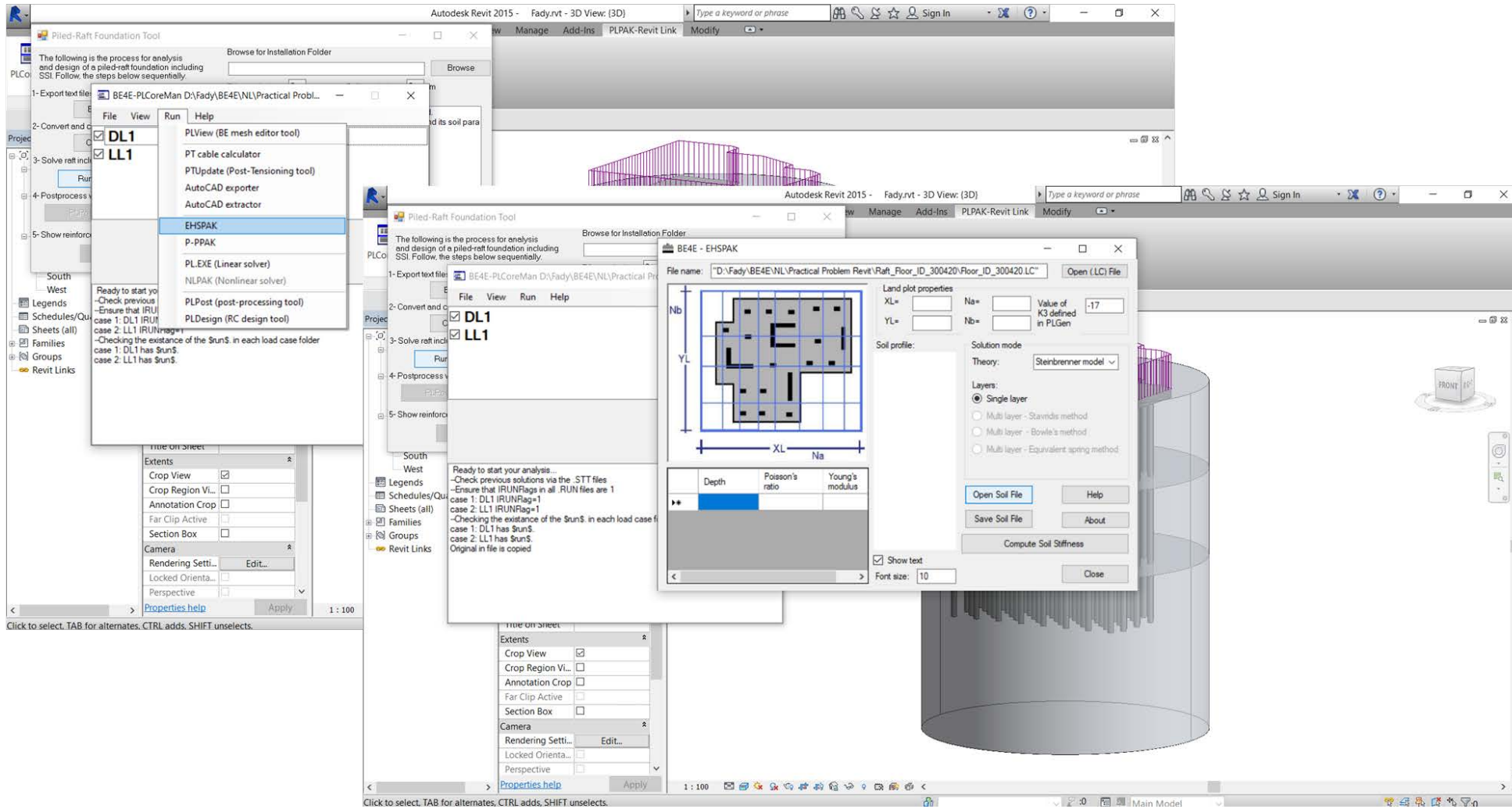
4. Foundation Package (Foundation)

Can open the Gen model of the piled raft from the raft folder (extracted from Revit)



4. Foundation Package (Foundation)

7- Run EHSPAK



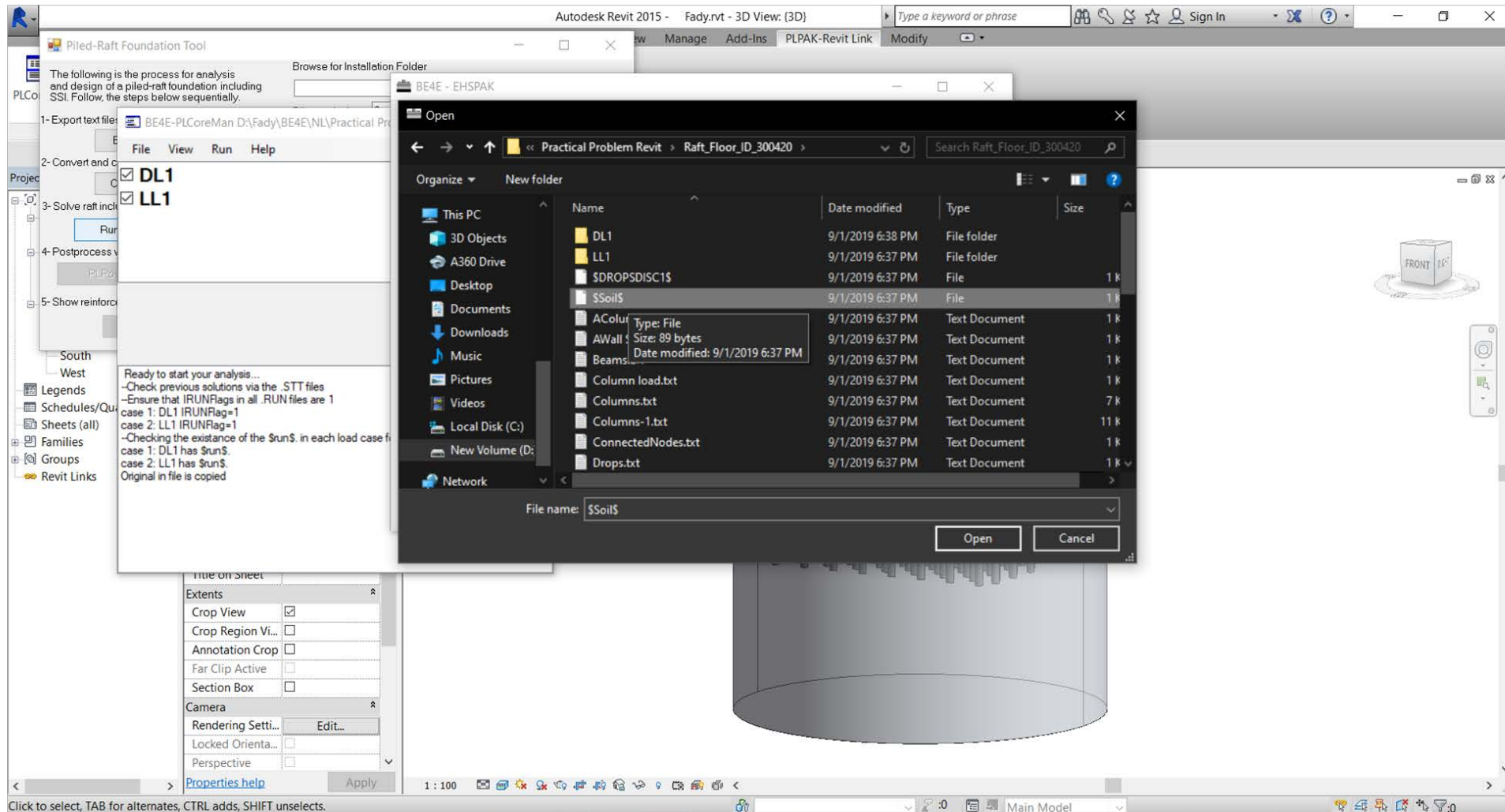
The screenshot displays the Piled-Raft Foundation Tool interface within Autodesk Revit 2015. The 'Run' menu is open, highlighting 'EHSPAK'. The 'EHSPAK' dialog box is active, showing the following details:

- File name:** "D:\Fady\BE4E\NL\Practical Problem Revit\Raft_Floor_ID_300420\Floor_ID_300420.LC"
- Land plot properties:** XL=, Na=, Value of K3 defined in PLGen: -17, YL=, Nb=
- Solution mode:** Steinbrenner model
- Layers:** Single layer, Multi layer - Stavids method, Multi layer - Bowle's method, Multi layer - Equivalent spring method
- Buttons:** Open Sol File, Save Sol File, Compute Soil Stiffness, Help, About, Close
- Other options:** Show text, Font size: 10

The background shows a 3D model of a foundation structure with a grid overlay. The software interface includes various toolbars and panels, such as the 'Project' panel on the left and the 'Properties' panel at the bottom.

4. Foundation Package (FoundraK)

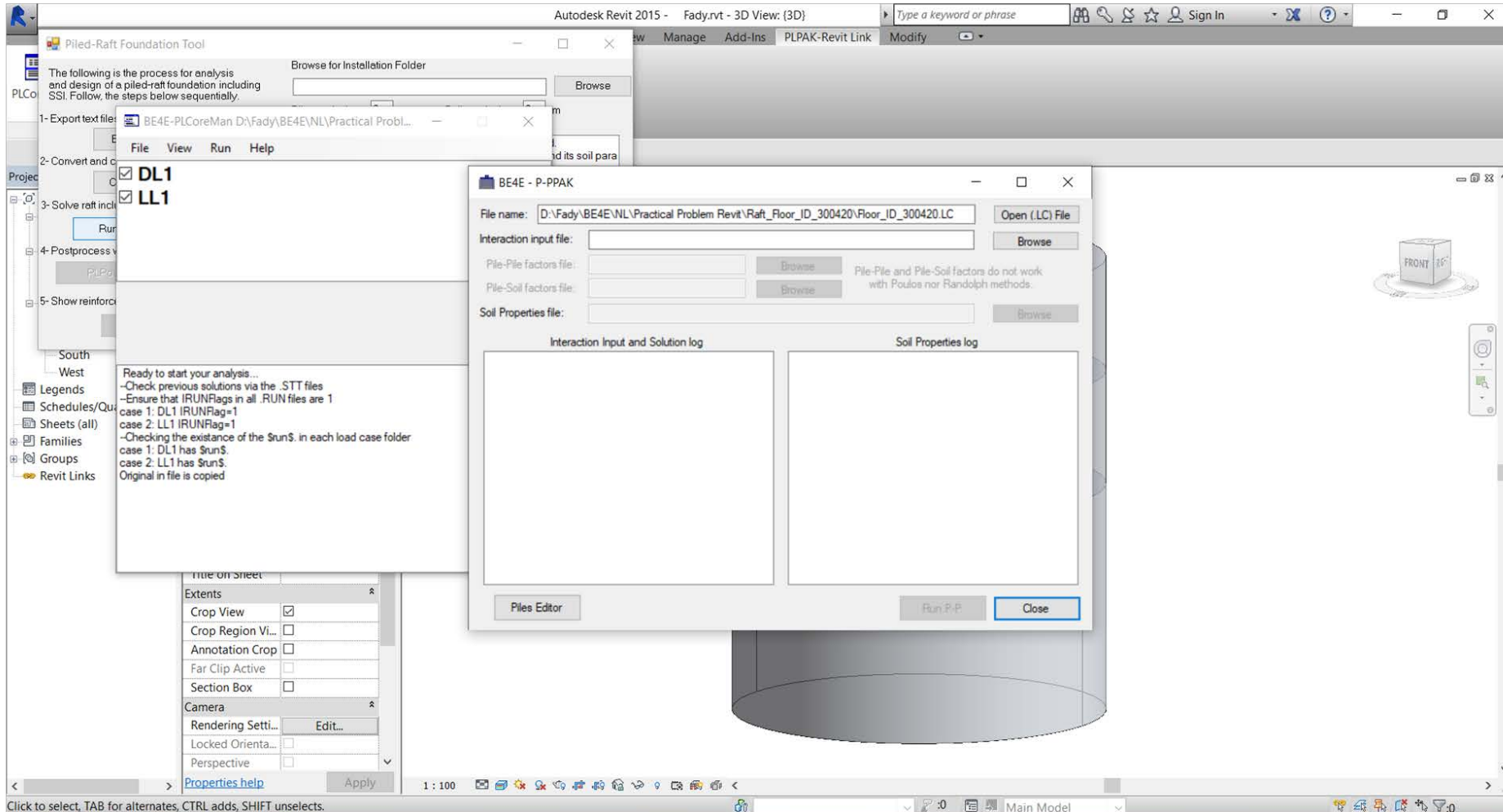
Browse \$Soil\$ in piled raft folder (automatically extracted from Revit)



The screenshot displays the Autodesk Revit 2015 interface with the Piled-Raft Foundation Tool open. The tool's main window shows a list of steps for analysis and design, including exporting text files, converting and solving the raft, and postprocessing. A 'Ready to start your analysis...' dialog box is visible, providing instructions on checking previous solutions and ensuring IRUNFlags are set correctly for DL1 and LL1 cases. A file explorer window is open, showing the contents of the 'Raft_Floor_ID_300420' folder, which includes folders for DL1 and LL1, and files for \$Soil\$, AColu, AWall, Beams, Column load.txt, Columns.txt, Columns-1.txt, ConnectedNodes.txt, and Drops.txt. The file explorer is currently displaying the \$Soil\$ file, and the 'File name' field is set to '\$Soil\$'. The Revit interface also shows a 3D view of a foundation structure and a Properties panel on the right side.

4. Foundation Package (Foundation)

8- Run P-PPAK



The screenshot displays the Autodesk Revit 2015 interface with the BE4E P-PPAK dialog box open. The dialog box is titled "BE4E - P-PPAK" and contains the following fields and options:

- File name: D:\Fady\BE4E\NL\Practical Problem Revit\Raft_Floor_ID_300420\Floor_ID_300420.LC (with "Open (LC) File" button)
- Interaction input file: (with "Browse" button)
- Pile-File factors file: (with "Browse" button)
- Pile-Soil factors file: (with "Browse" button)
- Soil Properties file: (with "Browse" button)

Below these fields are two empty text areas labeled "Interaction Input and Solution log" and "Soil Properties log". At the bottom of the dialog are "Files Editor", "Run P-P", and "Close" buttons.

In the background, the "Piled-Raft Foundation Tool" window is visible, showing a list of steps for analysis and design. A smaller window titled "BE4E-PLCoreMan" is also open, displaying a list of cases:

- DL1
- LL1

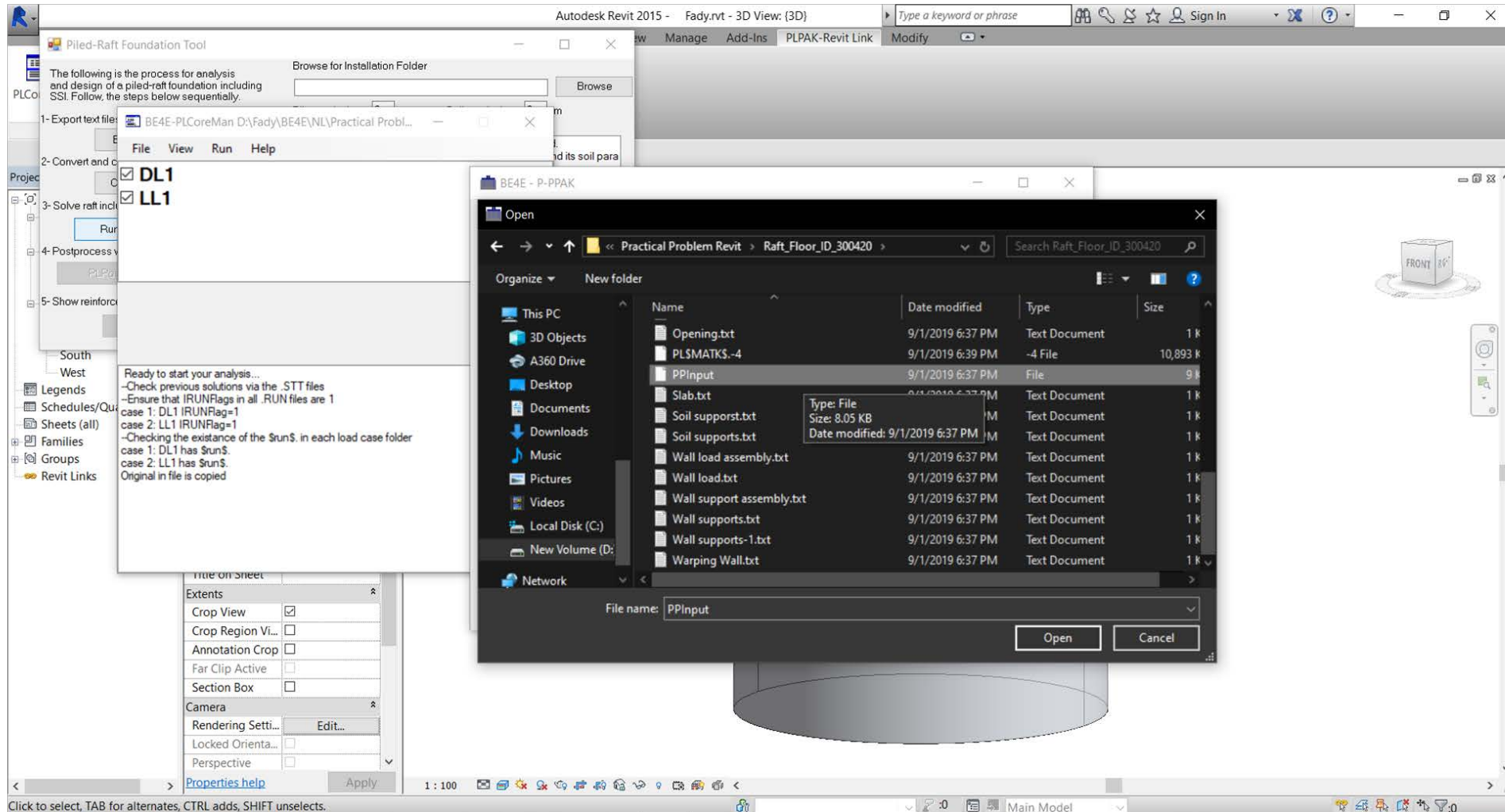
The "Ready to start your analysis..." window contains the following instructions:

- Check previous solutions via the .STT files
- Ensure that IRUNFlags in all .RUN files are 1
- case 1: DL1 IRUNFlag=1
- case 2: LL1 IRUNFlag=1
- Checking the existence of the \$run\$. in each load case folder
- case 1: DL1 has \$run\$.
- case 2: LL1 has \$run\$.
- Original in file is copied

The Revit interface also shows a "Properties" panel on the left with various settings for the current view, including "Crop View" (checked), "Crop Region Vi..." (unchecked), "Annotation Crop" (unchecked), "Far Clip Active" (unchecked), "Section Box" (unchecked), and "Camera" settings.

4. Foundation Package (FoundraK)

Browse PPIInput and \$Soil\$ in piled raft folder (automatically extracted from Revit)



The screenshot displays the Autodesk Revit 2015 interface with the Piled-Raft Foundation Tool open. The tool window shows a list of steps for analysis and design, including exporting text files, converting and solving raft inclusions, and postprocessing. A file explorer window is open, showing the contents of the 'Raft_Floor_ID_300420' folder, which includes files like 'Opening.txt', 'PLSMATKS--4', 'PPIInput', 'Slab.txt', 'Soil support.txt', 'Soil supports.txt', 'Wall load assembly.txt', 'Wall load.txt', 'Wall support assembly.txt', 'Wall supports.txt', 'Wall supports-1.txt', and 'Warping Wall.txt'. The 'PPIInput' file is selected, and its properties are shown: Type: File, Size: 8.05 KB, Date modified: 9/1/2019 6:37 PM. The Revit interface also shows a 'Ready to start your analysis...' dialog box with instructions for checking previous solutions and ensuring that IRUNFlags are set correctly for each load case. The Revit Properties palette is visible at the bottom, showing options for Extents, Annotation Crop, Far Clip Active, Section Box, Camera, and Rendering Settings.

Autodesk Revit 2015 - Fady.rvt - 3D View: (3D)

Piled-Raft Foundation Tool

The following is the process for analysis and design of a piled-raft foundation including SSI. Follow the steps below sequentially.

- 1- Export text files
- 2- Convert and solve raft inclusions
- 3- Solve raft inclusions
- 4- Postprocess view
- 5- Show reinforcement

File View Run Help

DL1
LL1

Ready to start your analysis...
-Check previous solutions via the .STT files
-Ensure that IRUNFlags in all .RUN files are 1
case 1: DL1 IRUNFlag=1
case 2: LL1 IRUNFlag=1
-Checking the existence of the \$Run\$. in each load case folder
case 1: DL1 has \$Run\$.
case 2: LL1 has \$Run\$.
Original in file is copied

Open

Practical Problem Revit > Raft_Floor_ID_300420

Name	Date modified	Type	Size
Opening.txt	9/1/2019 6:37 PM	Text Document	1 k
PLSMATKS--4	9/1/2019 6:39 PM	-4 File	10,893 k
PPIInput	9/1/2019 6:37 PM	File	9 k
Slab.txt	9/1/2019 6:37 PM	Text Document	1 k
Soil support.txt	9/1/2019 6:37 PM	Text Document	1 k
Soil supports.txt	9/1/2019 6:37 PM	Text Document	1 k
Wall load assembly.txt	9/1/2019 6:37 PM	Text Document	1 k
Wall load.txt	9/1/2019 6:37 PM	Text Document	1 k
Wall support assembly.txt	9/1/2019 6:37 PM	Text Document	1 k
Wall supports.txt	9/1/2019 6:37 PM	Text Document	1 k
Wall supports-1.txt	9/1/2019 6:37 PM	Text Document	1 k
Warping Wall.txt	9/1/2019 6:37 PM	Text Document	1 k

File name: PPIInput

Open Cancel

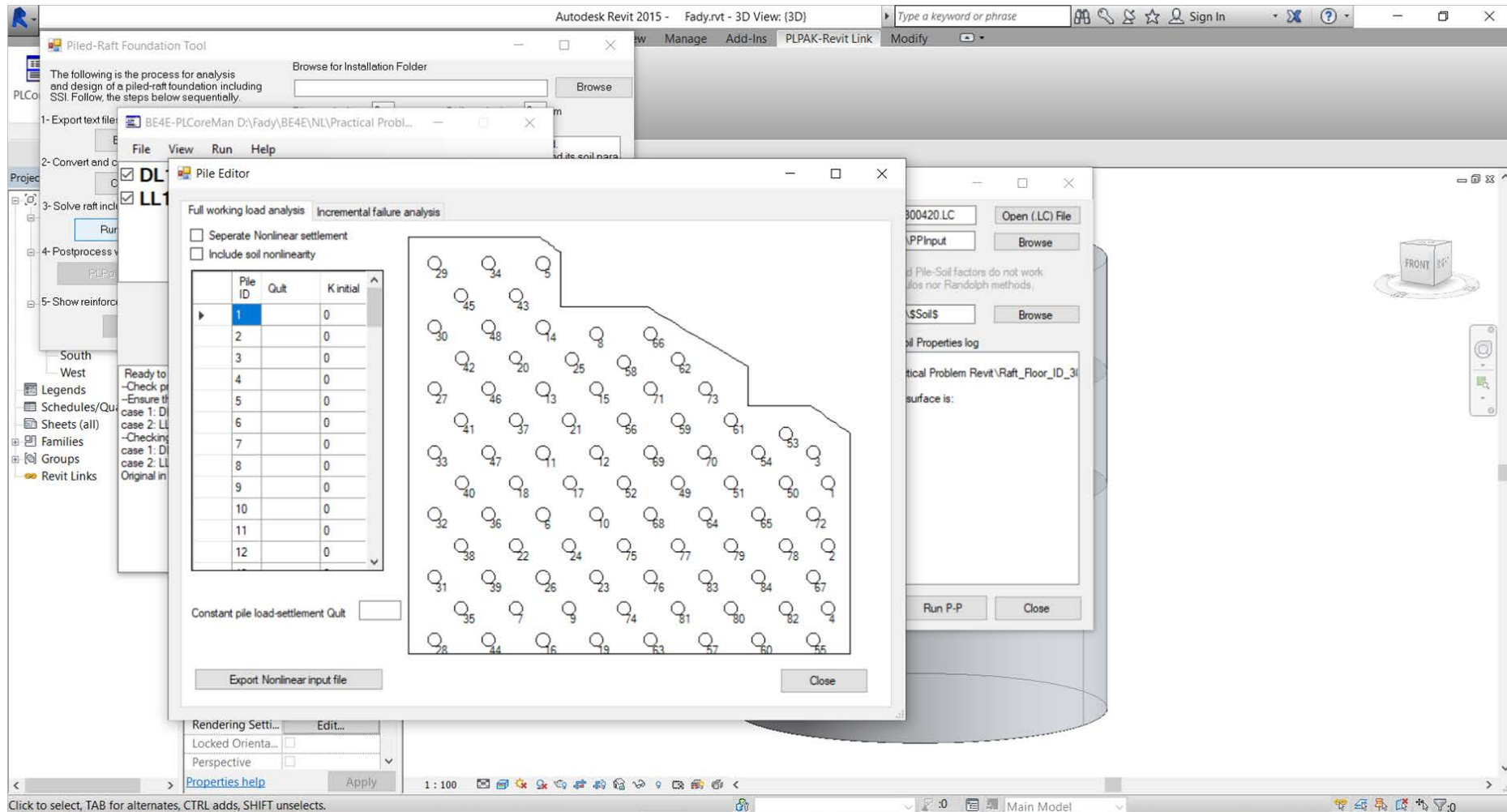
Click to select, TAB for alternates, CTRL adds, SHIFT unselects.

1 : 100

Main Model

4. Foundation Package (FoundraK)

9- Open the pile editor

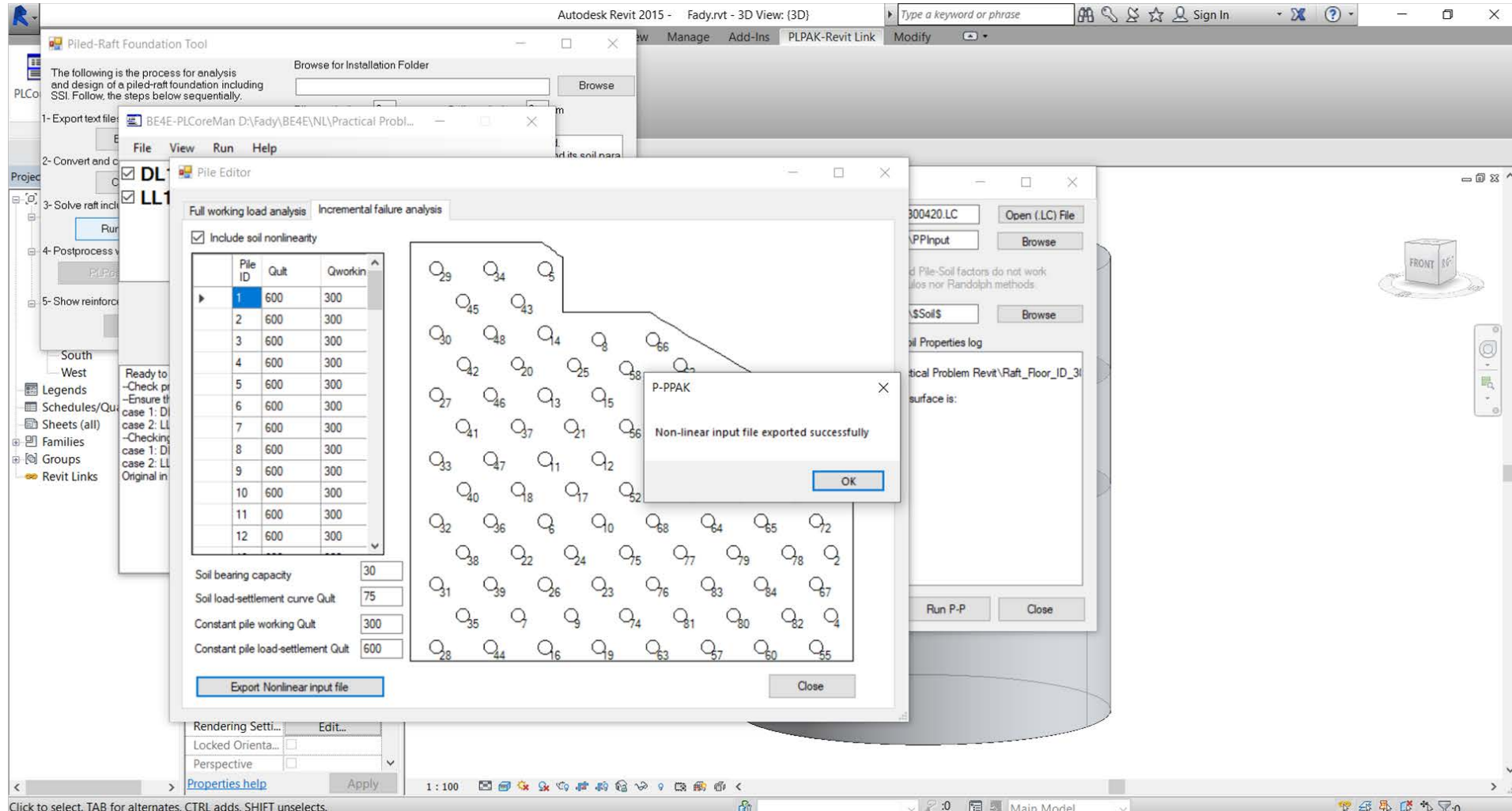


The screenshot displays the Autodesk Revit 2015 interface with the Piled-Raft Foundation Tool. The Pile Editor dialog box is open, showing the following data:

Pile ID	Quit	K initial
1	0	0
2	0	0
3	0	0
4	0	0
5	0	0
6	0	0
7	0	0
8	0	0
9	0	0
10	0	0
11	0	0
12	0	0

The dialog also includes a 3D view of the foundation layout with numbered piles (1-82) and various analysis options like 'Full working load analysis' and 'Incremental failure analysis'. Other windows visible include 'Browse for Installation Folder', 'Piled-Raft Foundation Tool' instructions, and 'Run P-P' dialog.

4. Foundation Package (FounderaK) Extract the NLinput file



The screenshot displays the 'Piled-Raft Foundation Tool' interface within Autodesk Revit 2015. The main window shows a 3D model of a foundation with a grid of numbered piles. A dialog box titled 'Full working load analysis' is open, featuring a table with the following data:

File ID	Quit	Qworkin
1	600	300
2	600	300
3	600	300
4	600	300
5	600	300
6	600	300
7	600	300
8	600	300
9	600	300
10	600	300
11	600	300
12	600	300

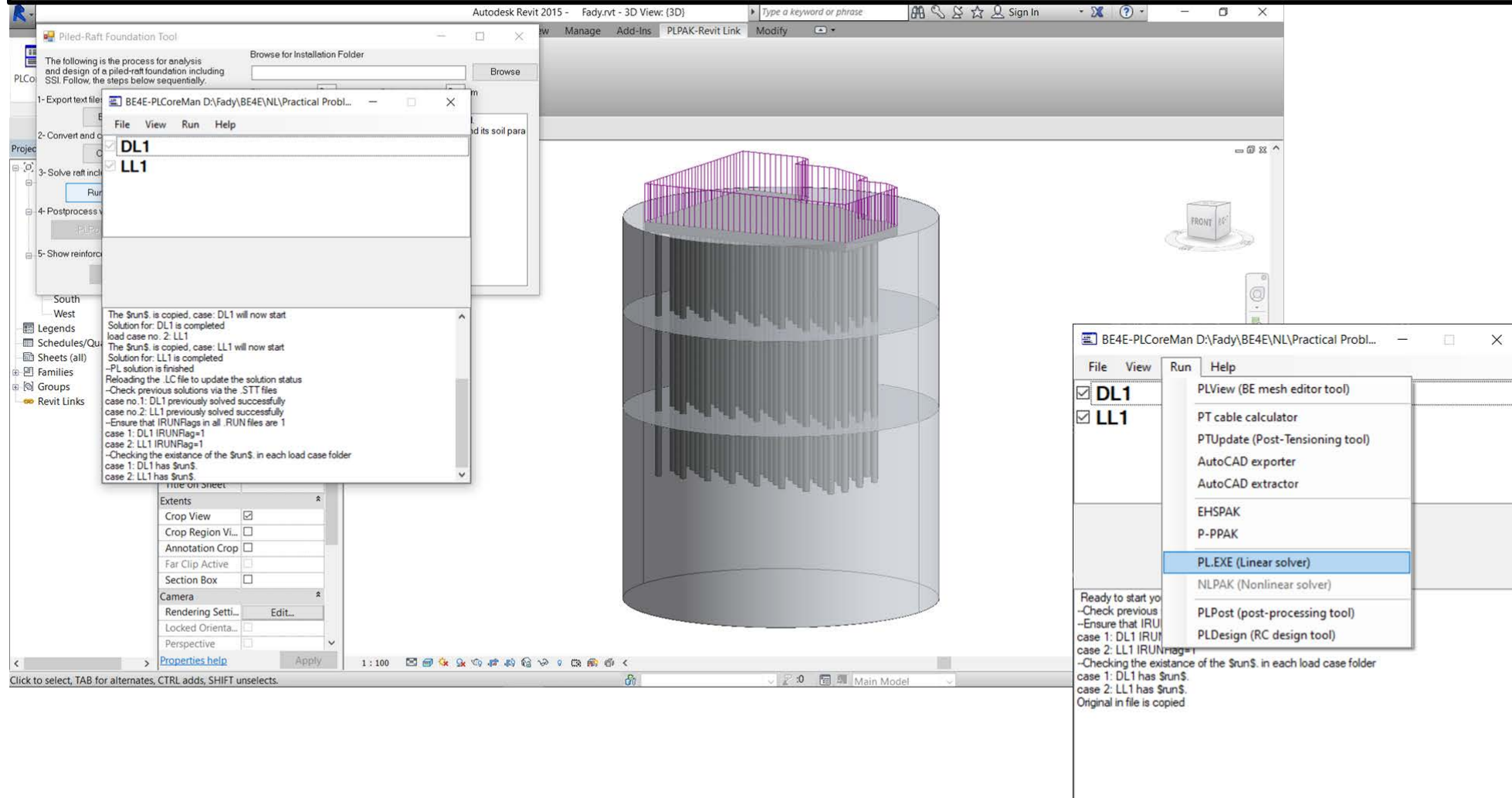
Below the table, the following parameters are set:

- Soil bearing capacity: 30
- Soil load-settlement curve Quit: 75
- Constant pile working Quit: 300
- Constant pile load-settlement Quit: 600

An 'Export Nonlinear input file' button is visible at the bottom of the dialog. A small 'P-PPAK' dialog box is also present, displaying the message: 'Non-linear input file exported successfully' with an 'OK' button.

4. Foundation Package (Foundations)

10- Run the linear solver PL.exe



The screenshot shows the Autodesk Revit 2015 interface with the Piled-Raft Foundation Tool. The main window displays a 3D model of a foundation with a grid of piles. A console window on the left shows the execution progress for cases DL1 and LL1. A 'Run' menu is open, highlighting 'PL.EXE (Linear solver)'. A status bar at the bottom indicates 'Main Model'.

Console Output:

```
The $run$. is copied, case: DL1 will now start  
Solution for: DL1 is completed  
load case no. 2: LL1  
The $run$. is copied, case: LL1 will now start  
Solution for: LL1 is completed  
-PL solution is finished  
Reloading the .LC file to update the solution status  
-Check previous solutions via the .STT files  
case no. 1: DL1 previously solved successfully  
case no. 2: LL1 previously solved successfully  
-Ensure that IRUNFlags in all .RUN files are 1  
case 1: DL1 IRUNFlag=1  
case 2: LL1 IRUNFlag=1  
-Checking the existence of the $run$. in each load case folder  
case 1: DL1 has $run$.  
case 2: LL1 has $run$.
```

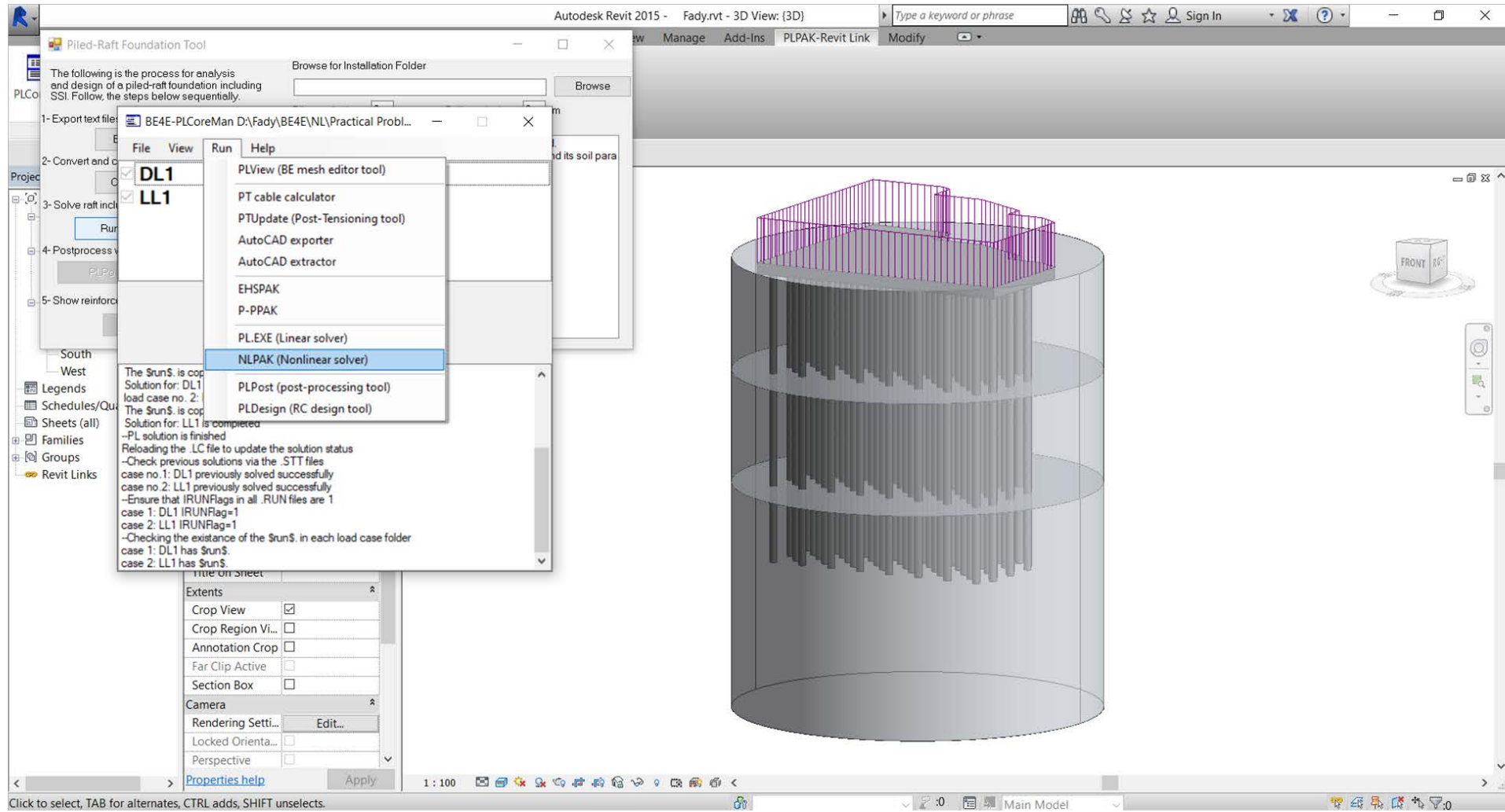
Run Menu Options:

- DL1
- LL1
- PLView (BE mesh editor tool)
- PT cable calculator
- PTUpdate (Post-Tensioning tool)
- AutoCAD exporter
- AutoCAD extractor
- EHSPAK
- P-PPAK
- PL.EXE (Linear solver)**
- NLPAK (Nonlinear solver)
- PLPost (post-processing tool)
- PLDesign (RC design tool)

Ready to start yo
-Check previous
-Ensure that IRU
case 1: DL1 IRU
case 2: LL1 IRUN
-Checking the existence of the \$run\$. in each load case folder
case 1: DL1 has \$run\$.
case 2: LL1 has \$run\$.
Original in file is copied

4. Foundation Package (FoundationK)

11- Run the nonlinear solver NLPK



The screenshot displays the Autodesk Revit 2015 interface with the Piled-Raft Foundation Tool. The tool window is open, showing a list of steps for analysis and design. A context menu is open over the 'Run' button, with 'NLPK (Nonlinear solver)' selected. The 3D view shows a cylindrical foundation structure with a grid of reinforcement bars. The status bar at the bottom indicates '1 : 100' and 'Main Model'.

The Piled-Raft Foundation Tool window contains the following text:

The following is the process for analysis and design of a piled-raft foundation including SSI. Follow the steps below sequentially.

- 1- Export text files
- 2- Convert and...
- 3- Solve raft incl...
- 4- Postprocess v...
- 5- Show reinforce...

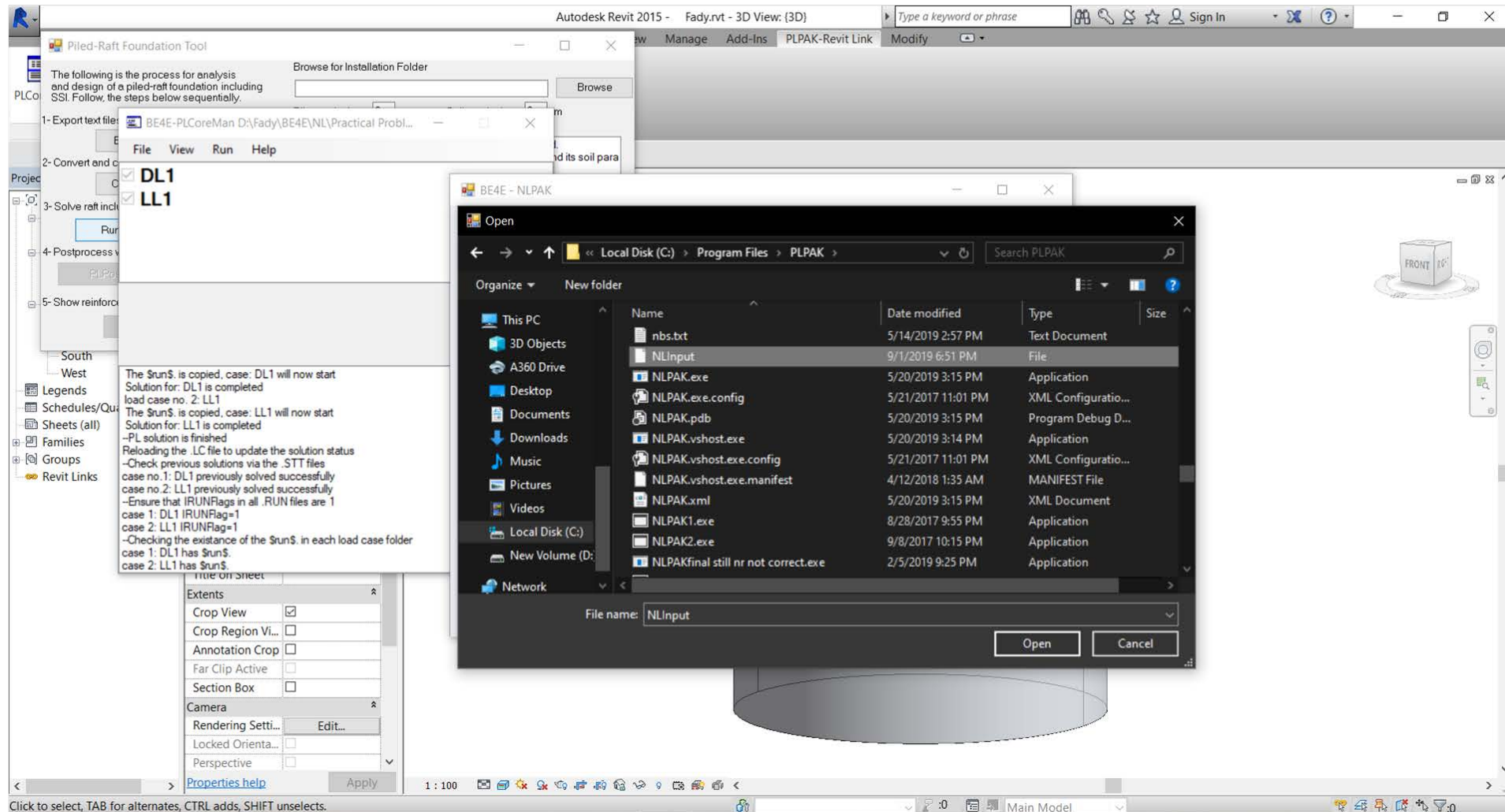
The context menu includes the following options:

- PLView (BE mesh editor tool)
- PT cable calculator
- PTUpdate (Post-Tensioning tool)
- AutoCAD exporter
- AutoCAD extractor
- EHSPAK
- P-PPAK
- PL.EXE (Linear solver)
- NLPK (Nonlinear solver)**
- PLPost (post-processing tool)
- PLDesign (RC design tool)

The status bar at the bottom of the Revit window shows '1 : 100' and 'Main Model'.

4. Foundation Package (FoundraK)

Browse NInput in PLPAK folder (extracted from pile editor)

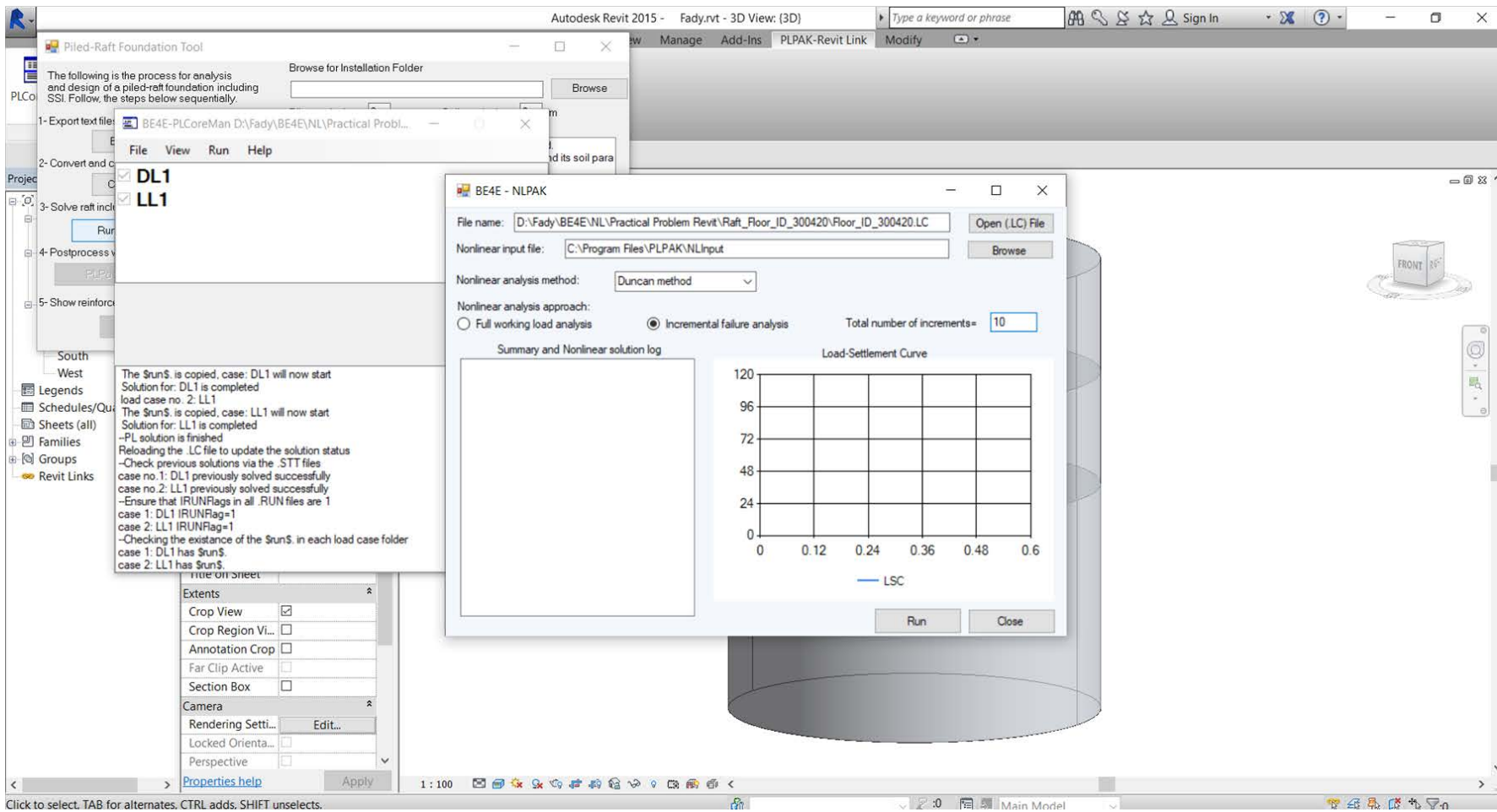


The screenshot displays the Autodesk Revit 2015 interface with the Piled-Raft Foundation Tool. The tool's main window shows a list of load cases: DL1 and LL1. A text box provides instructions for running the tool, including steps for copying the \$run\$. file and checking the existence of the \$run\$. file in each load case folder. A file explorer window is open, showing the contents of the PLPAK folder, with the NInput file selected. The file explorer window shows the following files:

Name	Date modified	Type	Size
nbs.txt	5/14/2019 2:57 PM	Text Document	
NInput	9/1/2019 6:51 PM	File	
NLPAK.exe	5/20/2019 3:15 PM	Application	
NLPAK.exe.config	5/21/2017 11:01 PM	XML Configuratio...	
NLPAK.pdb	5/20/2019 3:15 PM	Program Debug D...	
NLPAK.vshost.exe	5/20/2019 3:14 PM	Application	
NLPAK.vshost.exe.config	5/21/2017 11:01 PM	XML Configuratio...	
NLPAK.vshost.exe.manifest	4/12/2018 1:35 AM	MANIFEST File	
NLPAK.xml	5/20/2019 3:15 PM	XML Document	
NLPAK1.exe	8/28/2017 9:55 PM	Application	
NLPAK2.exe	9/8/2017 10:15 PM	Application	
NLPAKfinal still nr not correct.exe	2/5/2019 9:25 PM	Application	

4. Foundation Package (Foundations)

12- Choose your nonlinear approach: Full working load, or Incremental failure analysis (Input number of increments)



The screenshot displays the Autodesk Revit 2015 interface with the BE4E-NLPAK dialog box open. The dialog box is titled "BE4E - NLPAK" and contains the following configuration options:

- File name:** D:\Fady\BE4E\NL\Practical Problem Revit\Raft_Floor_ID_300420\Floor_ID_300420.LC
- Nonlinear input file:** C:\Program Files\PLPAK\NLinput
- Nonlinear analysis method:** Duncan method
- Nonlinear analysis approach:**
 - Full working load analysis
 - Incremental failure analysis
- Total number of increments:** 10

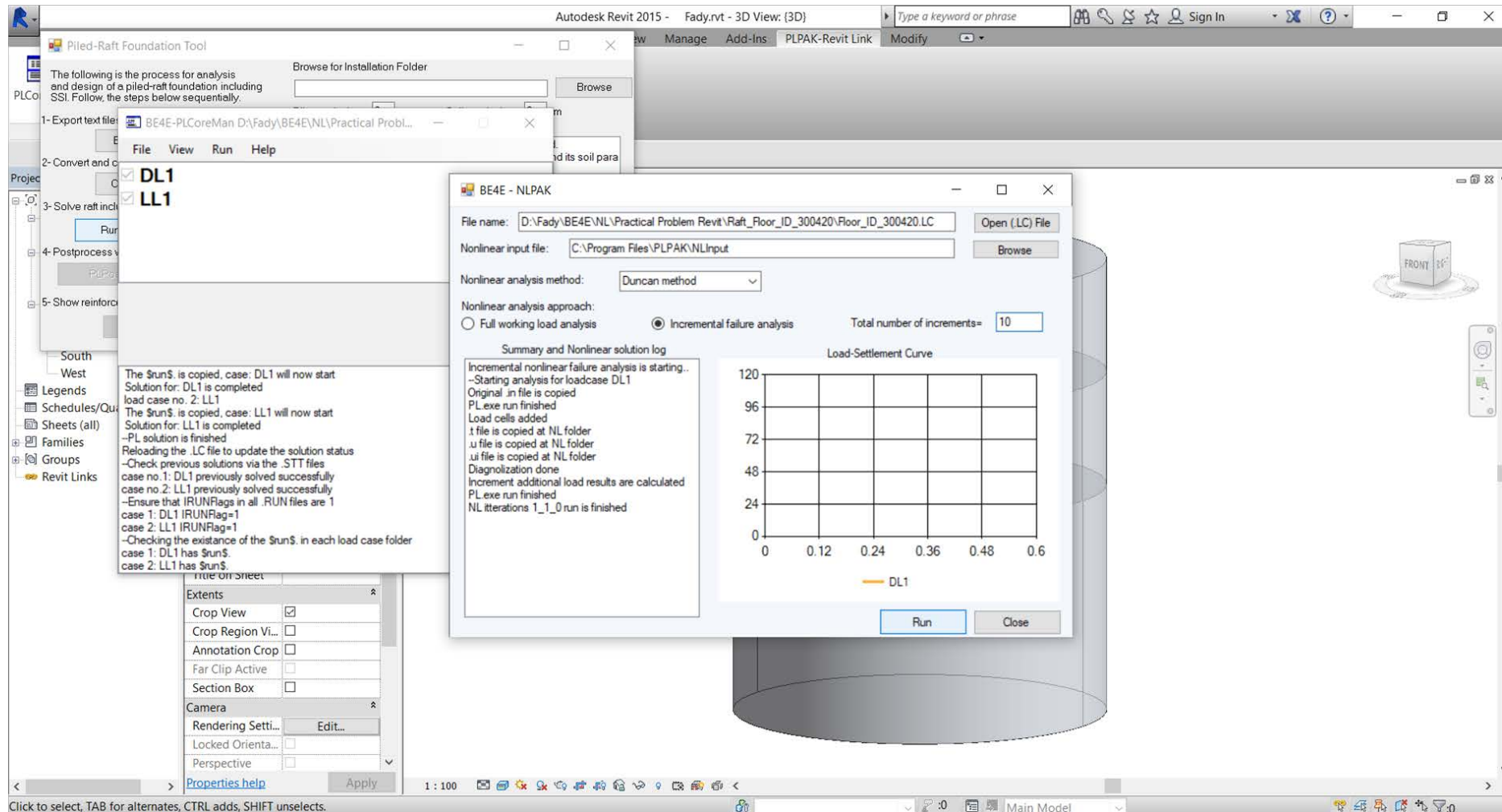
Below the configuration options, there are two panels:

- Summary and Nonlinear solution log:** A text area for logging the analysis process.
- Load-Settlement Curve:** A graph showing the relationship between load and settlement. The y-axis ranges from 0 to 120, and the x-axis ranges from 0 to 0.6. A legend indicates "LSC" (Load-Settlement Curve).

The background shows the Revit interface with a 3D view of a foundation and various toolbars.

4. Foundation Package (Foundation)

13- Run the solver



The screenshot displays the Autodesk Revit 2015 interface with the Piled-Raft Foundation Tool and the BE4E - NLPK dialog box open. The tool window shows a list of steps for analysis and design, with 'DL1' and 'LL1' selected. The BE4E - NLPK dialog box is configured with the following settings:

- File name: D:\Fady\BE4E\NL\Practical Problem Revit\Floor_ID_300420\Floor_ID_300420.LC
- Nonlinear input file: C:\Program Files\PLPAK\NLInput
- Nonlinear analysis method: Duncan method
- Nonlinear analysis approach: Full working load analysis, Incremental failure analysis
- Total number of increments: 10

The dialog box also displays a 'Summary and Nonlinear solution log' and a 'Load-Settlement Curve' graph. The log shows the following steps:

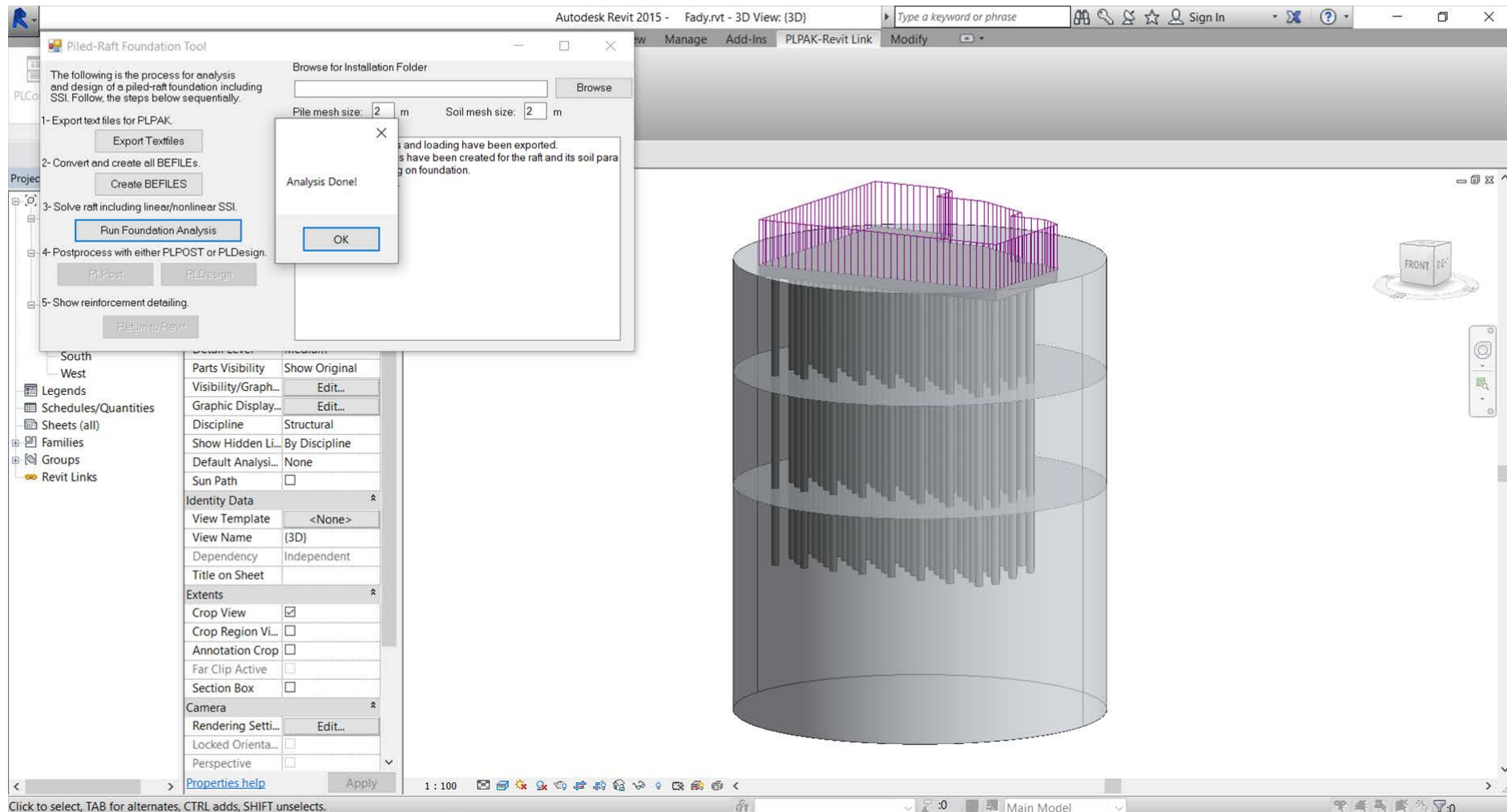
```

Incremental nonlinear failure analysis is starting..
--Starting analysis for loadcase DL1
Original .in file is copied
PL exe run finished
Load cells added
.i file is copied at NL folder
.u file is copied at NL folder
.ui file is copied at NL folder
Diagnolization done
Increment additional load results are calculated
PL exe run finished
NL iterations 1_1_0_run is finished
    
```

The graph shows a single data series for 'DL1' with a y-axis ranging from 0 to 120 and an x-axis ranging from 0 to 0.6. The 'Run' button is highlighted in blue.

4. Foundation Package (Foundations)

14- Close PLCoreMan and go back to the wizard (Analysis done!)



The screenshot displays the Autodesk Revit 2015 interface. The main window shows a 3D view of a foundation model with a purple wireframe structure on top. The 'Piled-Raft Foundation Tool' wizard is open, showing a list of steps: 1- Export text files for PLPAK, 2- Convert and create all BEFILES, 3- Solve raft including linear/nonlinear SSI, 4- Postprocess with either PLPOST or PLDesign, and 5- Show reinforcement detailing. A dialog box titled 'Analysis Done!' is displayed over the wizard, indicating that the analysis is complete. The 'Properties' panel on the left shows various settings for the selected element, including 'Parts Visibility', 'Visibility/Graph...', 'Graphic Display...', 'Discipline', 'Show Hidden Li...', 'Default Analysi...', 'Sun Path', 'Identity Data', 'Extents', and 'Camera'.

4. Foundation Package (Foundations)

15- Open PLPost to show the results

The screenshot displays the Autodesk Revit 2015 interface. The main window shows a 3D view of a foundation structure with a grid of piles extending downwards from a raft. The 'Piled-Raft Foundation Tool' dialog box is open, showing the following steps:

- 1- Export text files for PLPAK. (Export Textfiles button)
- 2- Convert and create all BEFILES. (Create BEFILES button)
- 3- Solve raft including linear/nonlinear SSI. (Run Foundation Analysis button)
- 4- Postprocess with either PLPOST or PLDesign. (PLPost and PLDesign buttons)
- 5- Show reinforcement detailing. (Return to Revit button)

The dialog box also includes a 'Browse for Installation Folder' section with a 'Browse' button, and a 'Log of actions' section with the following text:

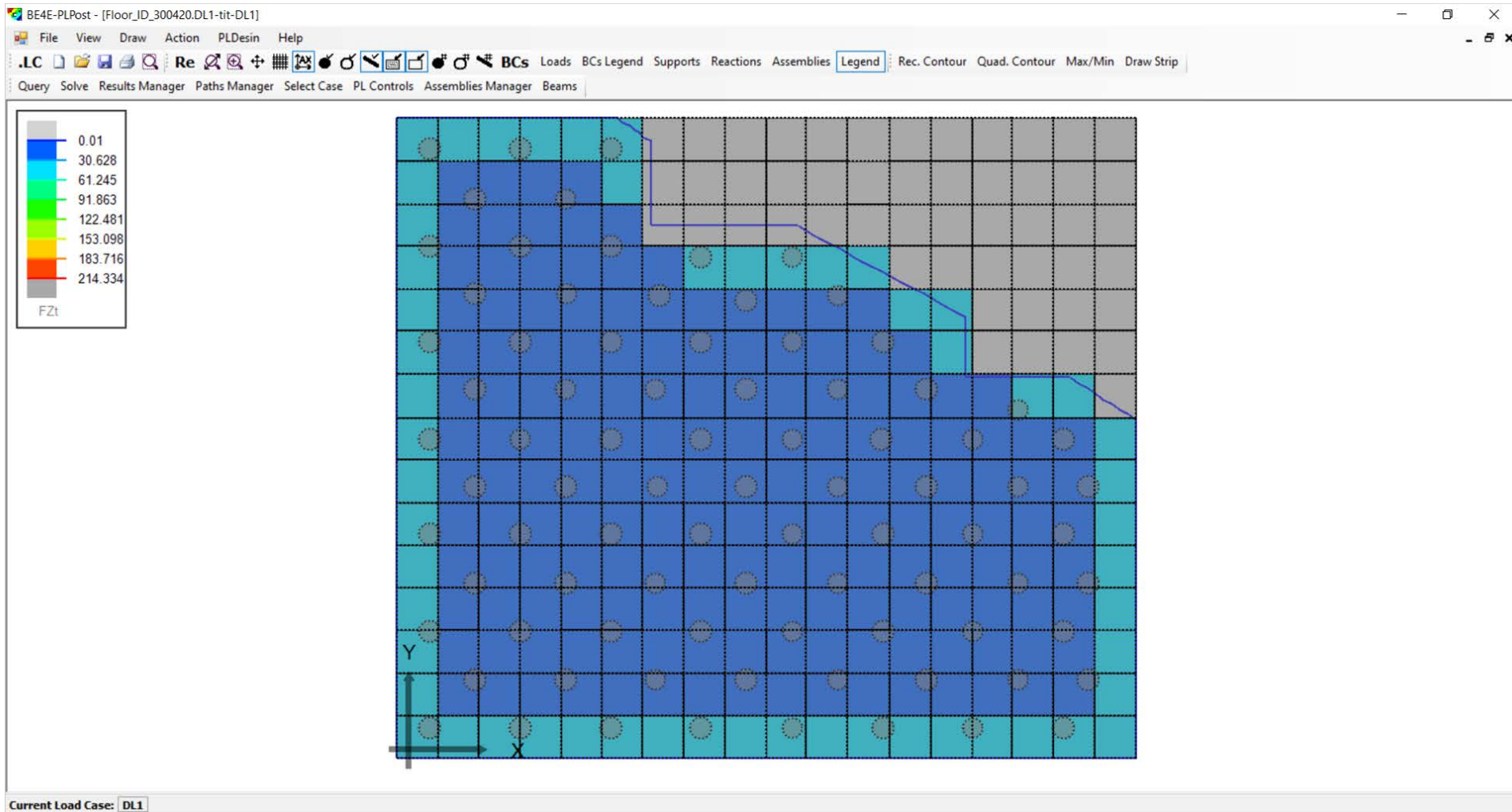
```
Structural components and loading have been exported.
Boundary element files have been created for the raft and its soil para
Step 3 chosen, solving on foundation.
Solving, please wait...
```

The Properties palette on the left shows the following settings:

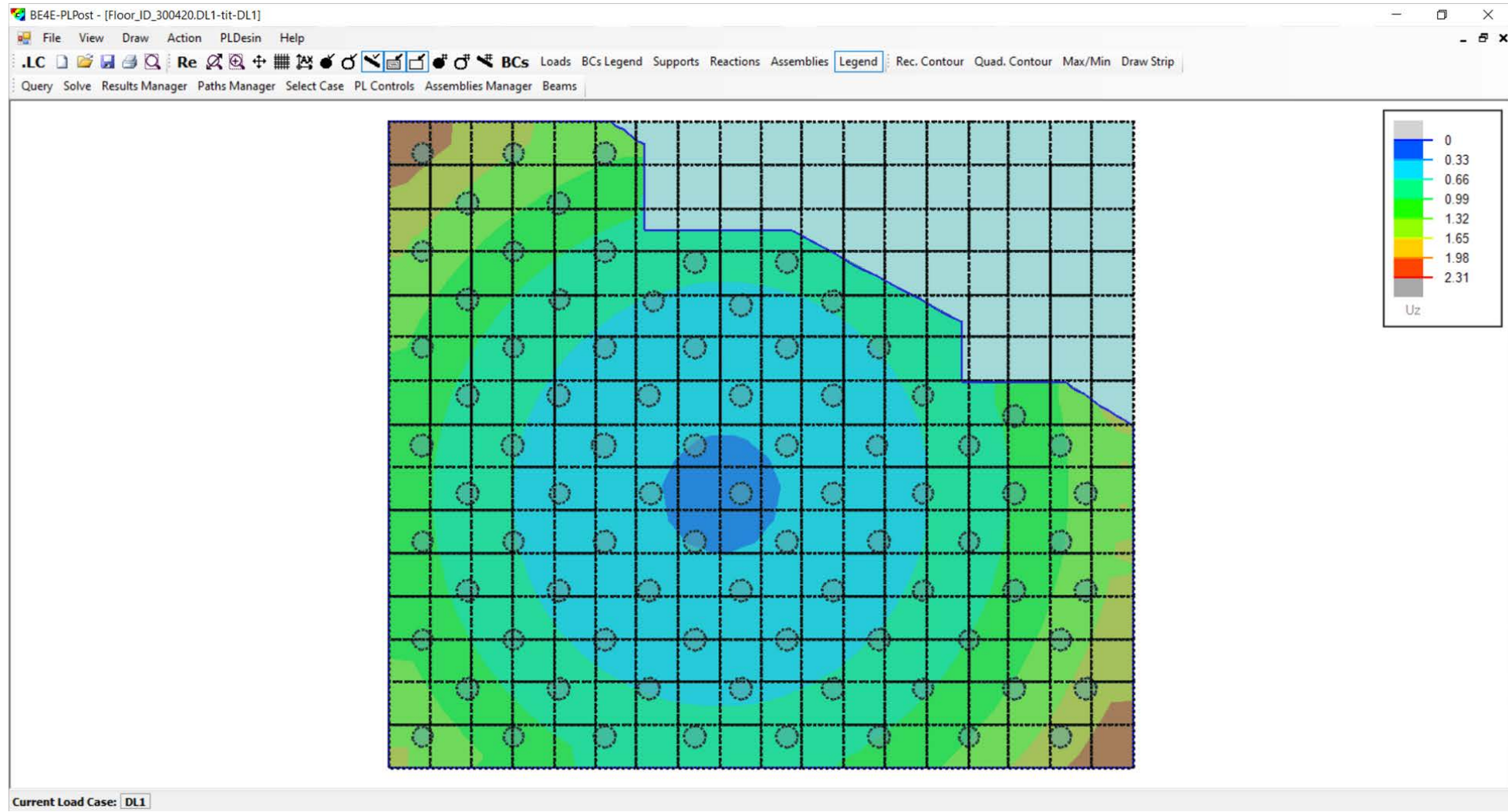
- Parts Visibility: Show Original
- Visibility/Graph...: Edit...
- Graphic Display...: Edit...
- Discipline: Structural
- Show Hidden Li...: By Discipline
- Default Analysi...: None
- Sun Path:
- Identity Data:
 - View Template: <None>
 - View Name: (3D)
 - Dependency: Independent
 - Title on Sheet:
- Extents:
 - Crop View:
 - Crop Region Vi...:
 - Annotation Crop:
 - Far Clip Active:
 - Section Box:
- Camera:
 - Rendering Setti...: Edit...
 - Locked Orienta...:
 - Perspective:

The bottom status bar shows '1 : 100' and 'Main Model'.

4. Foundation Package (Foundation) Soil and piles nonlinear reactions

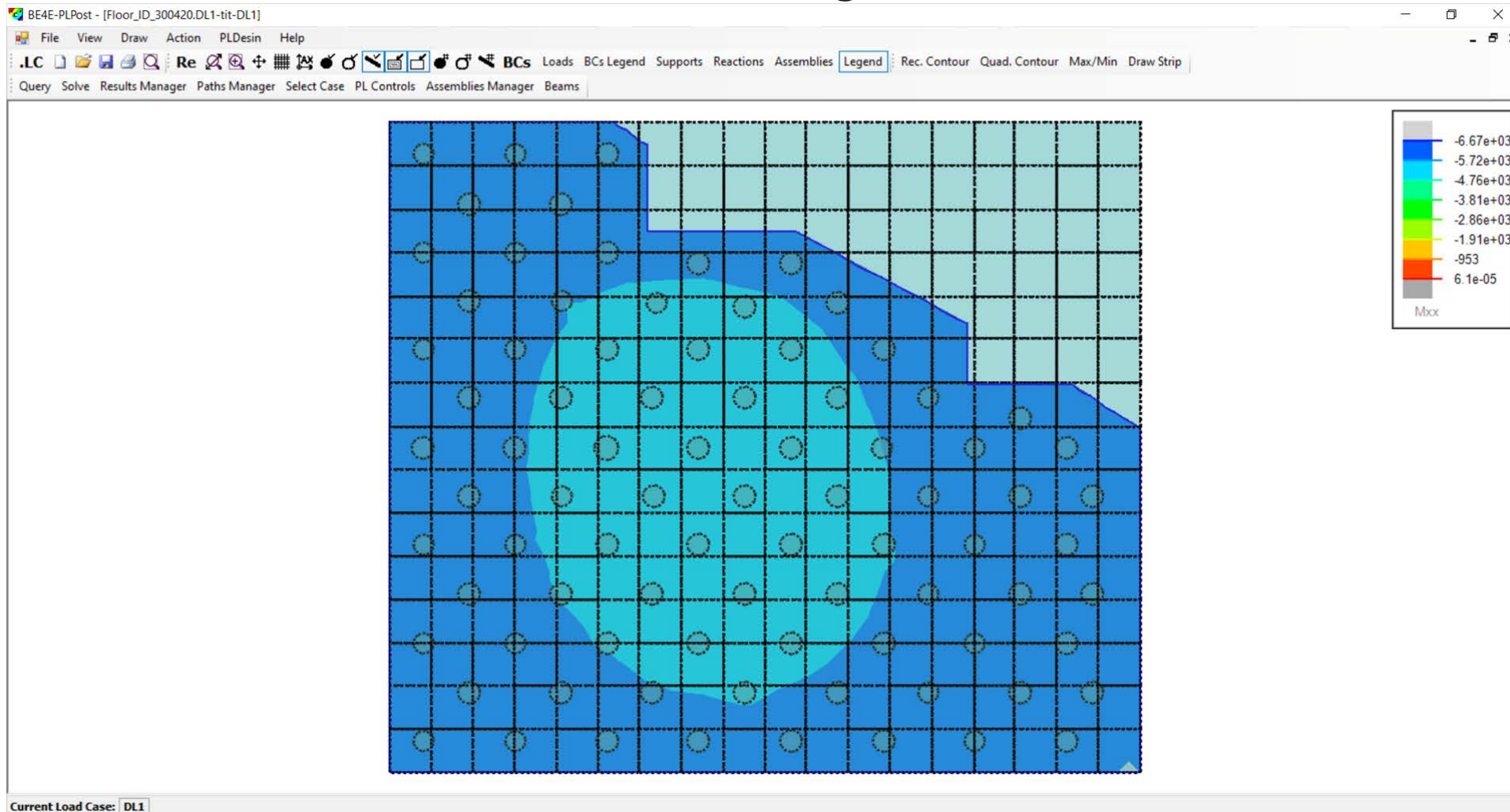


4. Foundation Package (Foundation) www.be4e.com Piled raft nonlinear settlement contours



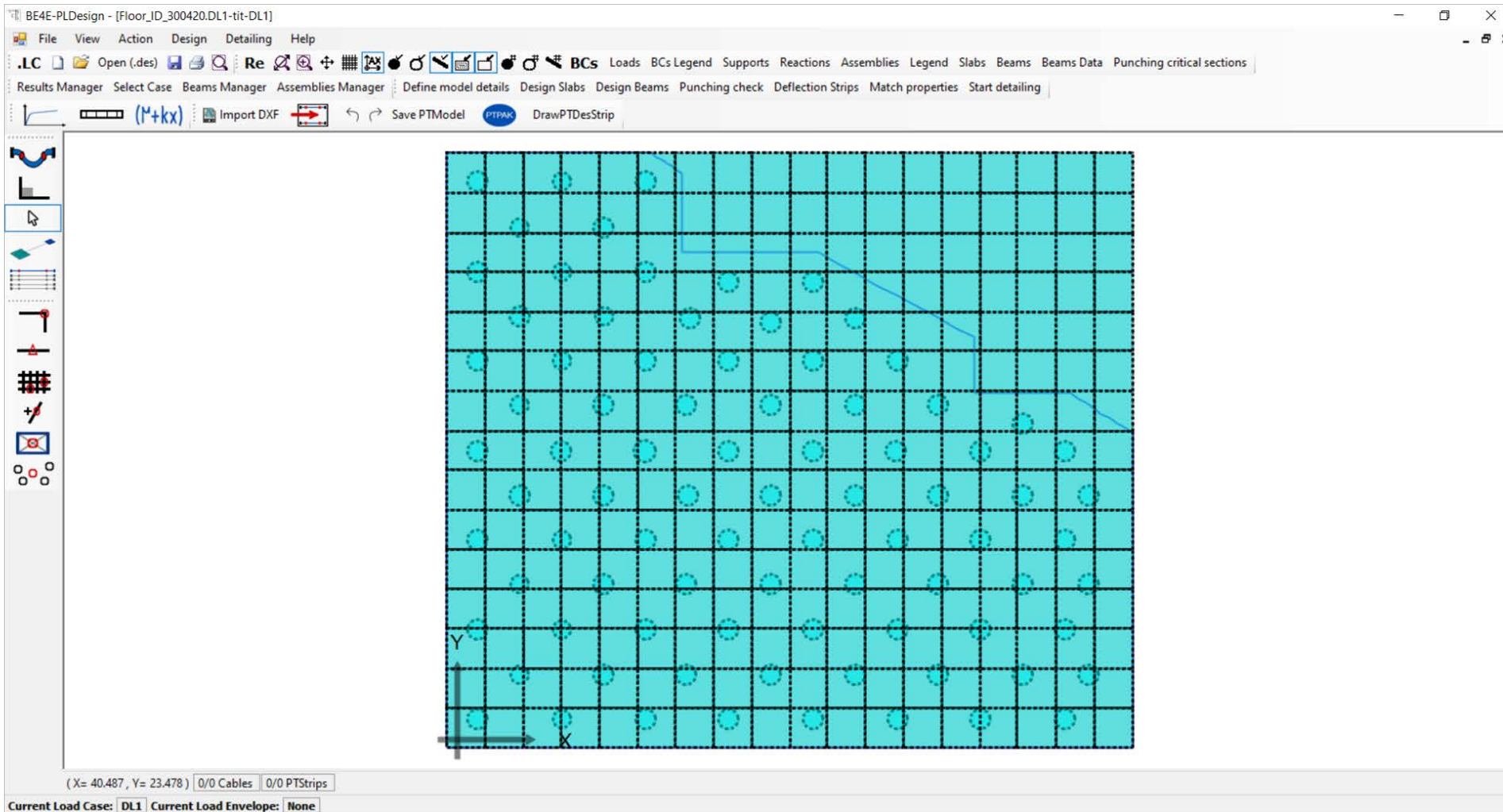
4. Foundation Package (Foundation)

Piled raft nonlinear bending moment contours

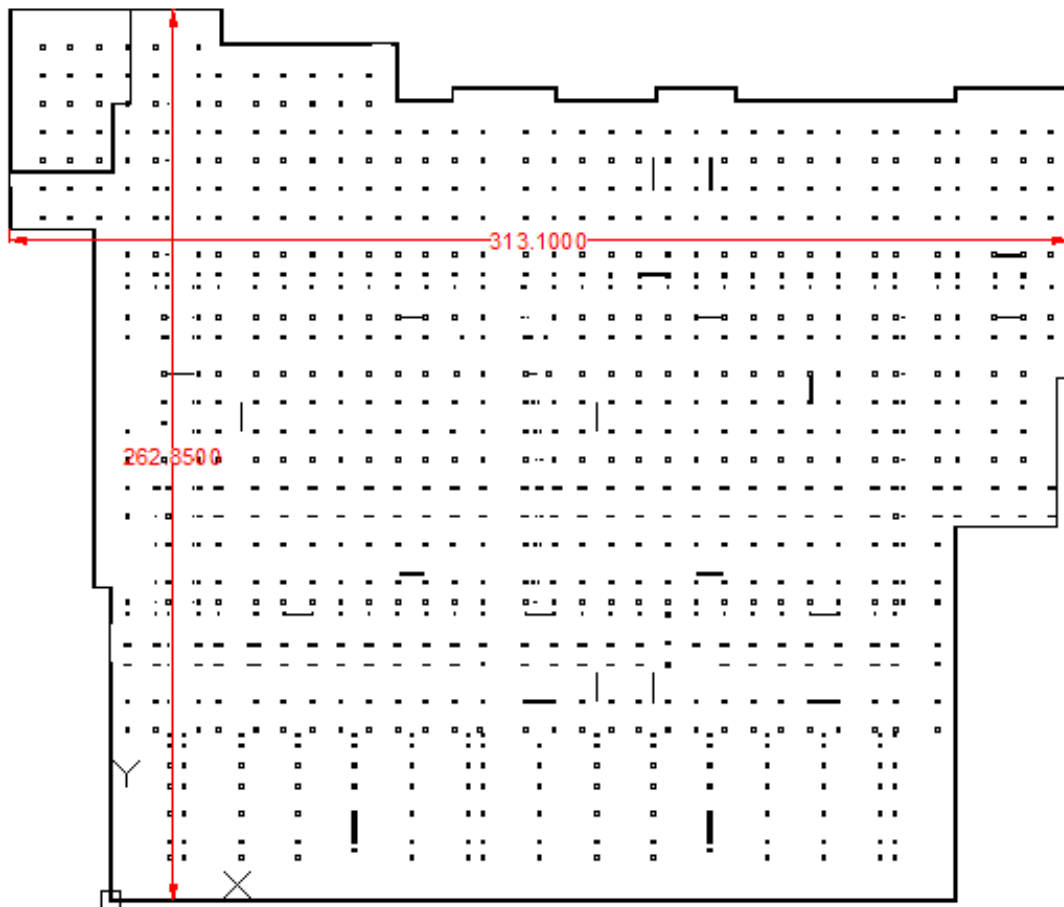


4. Foundation Package (Foundation)

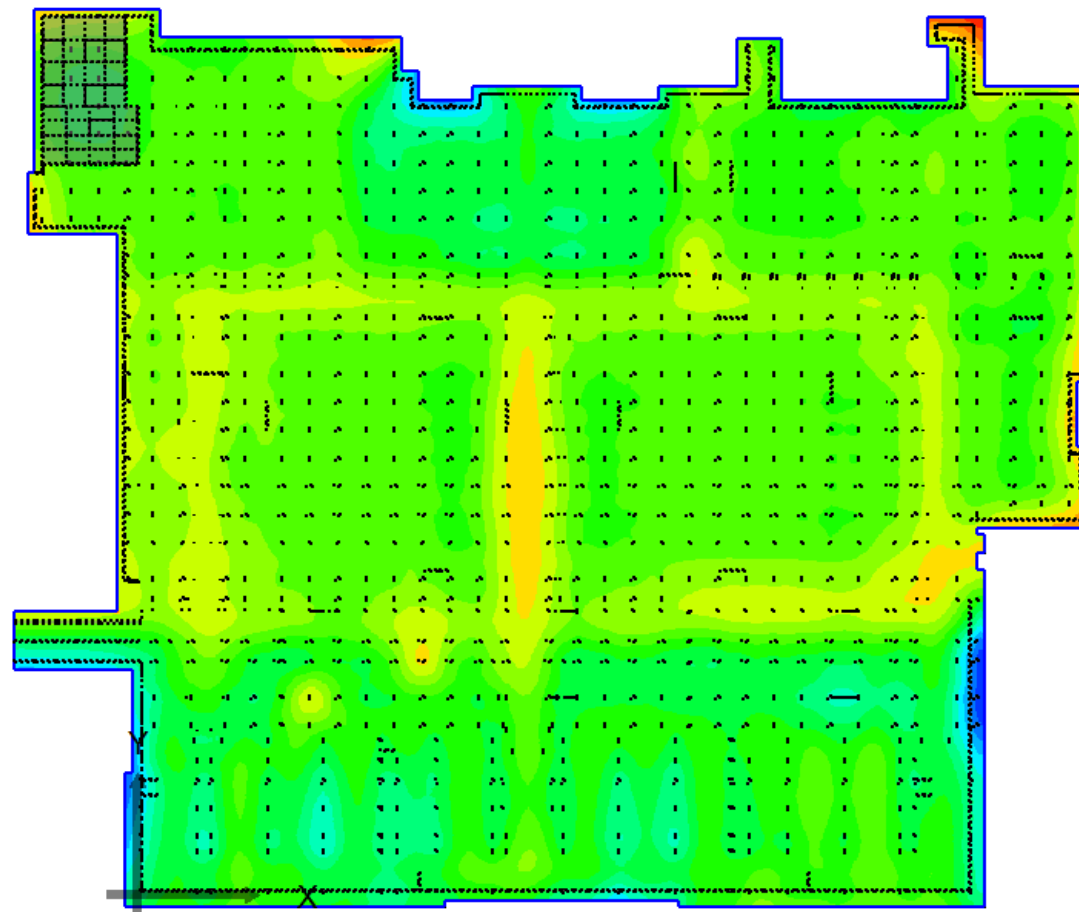
16- Open PLDesign to design the raft as RC or PT raft



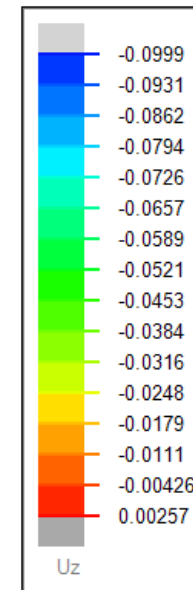
4. Foundation Package (Foundation)



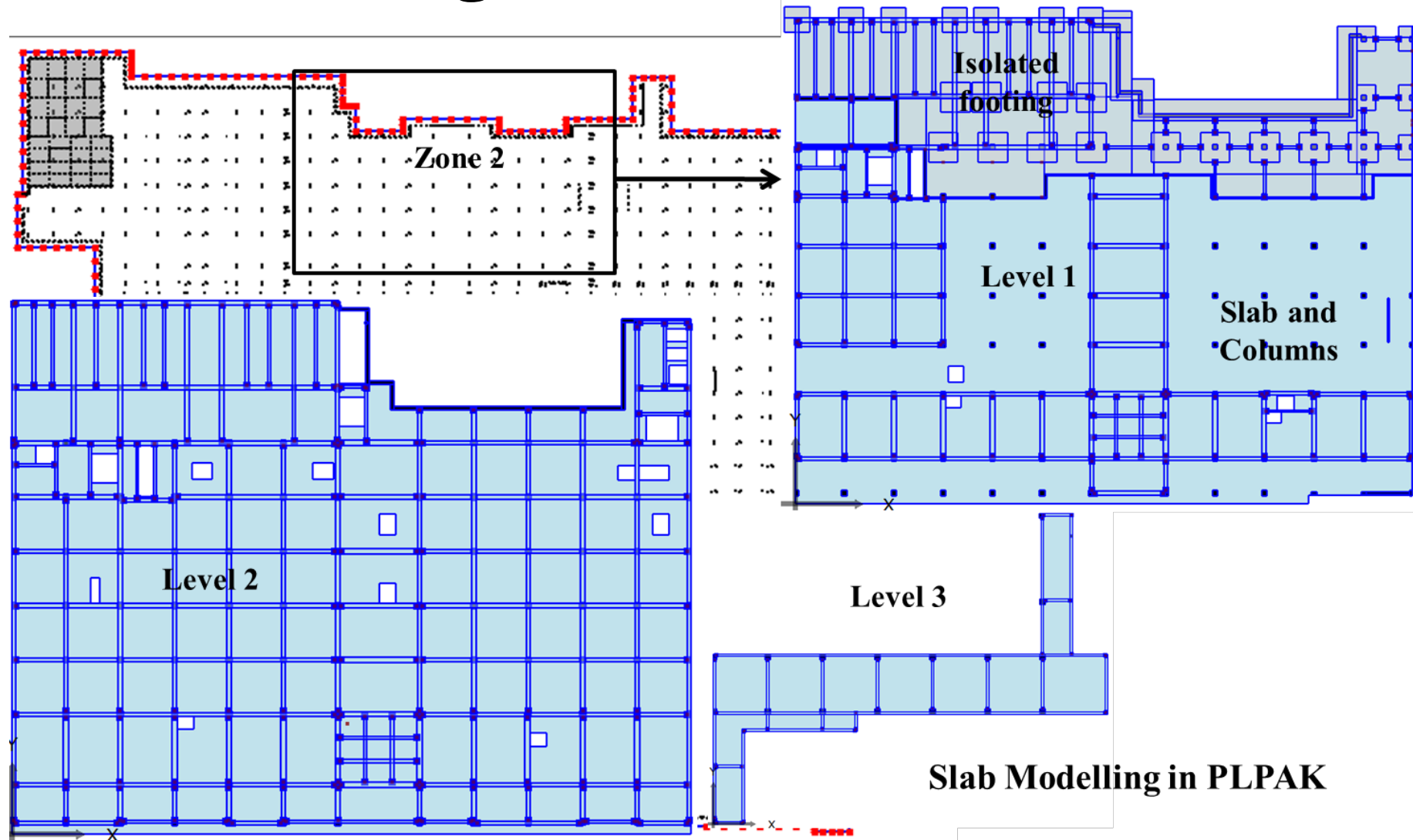
AutoCAD Drawing



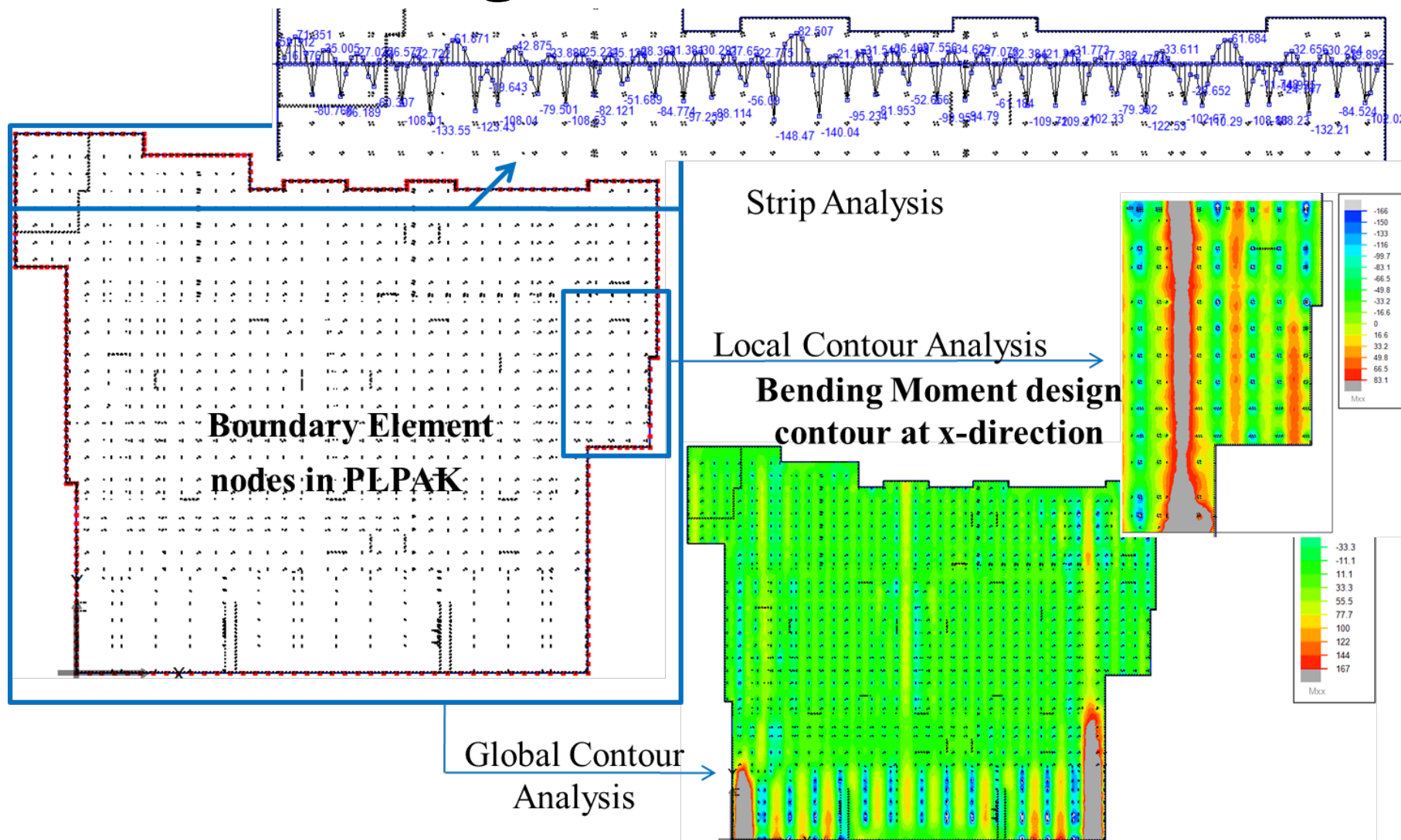
Deformation contour



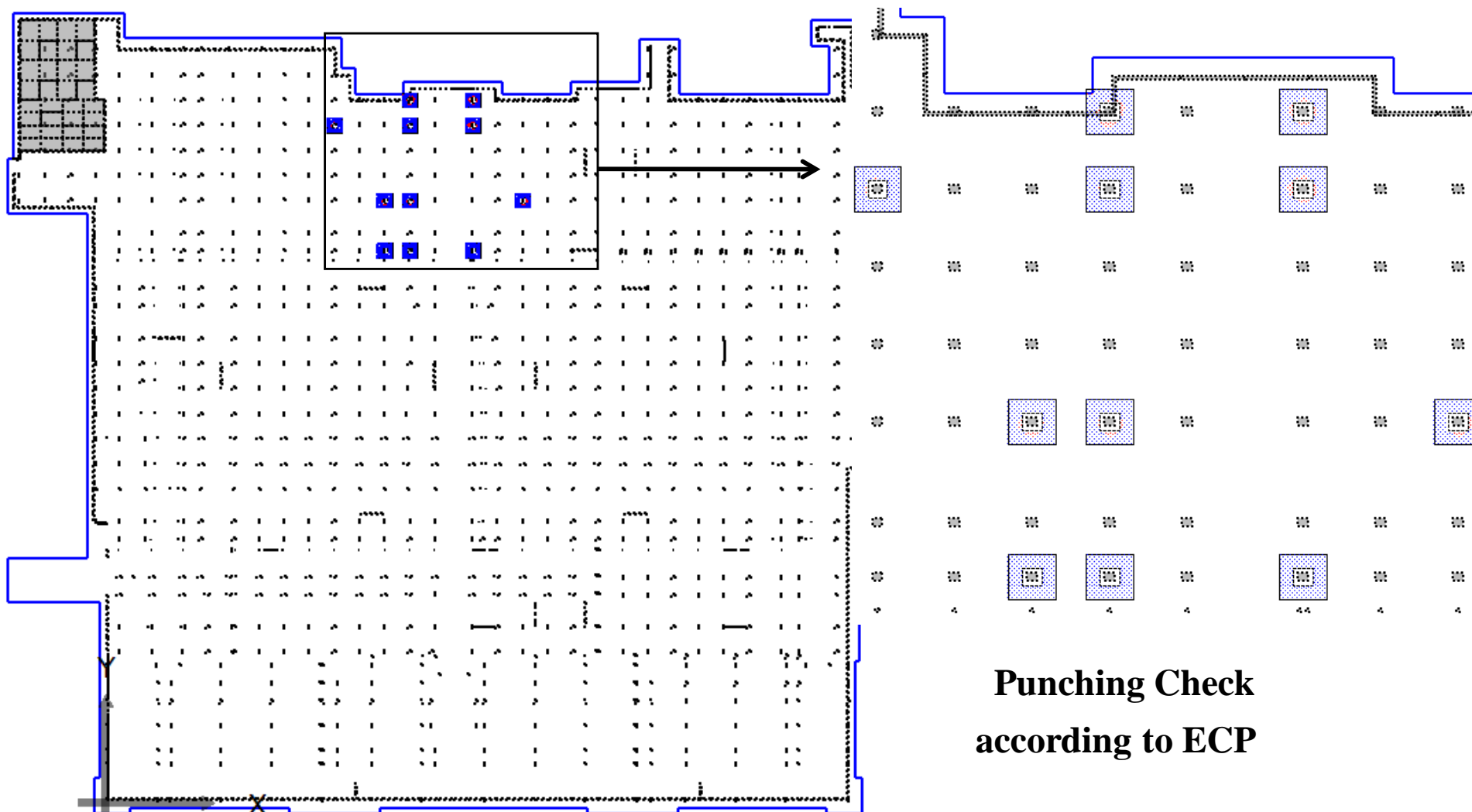
4. Foundation Package (Foundation)



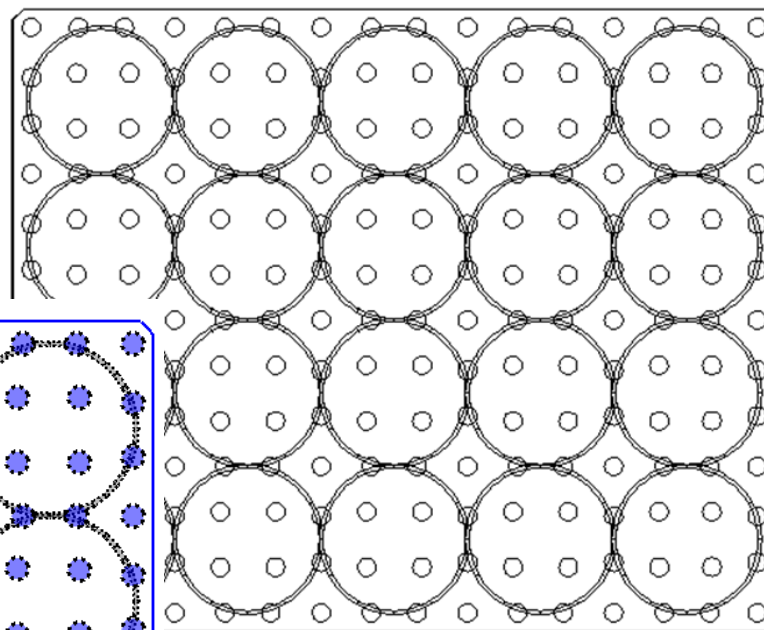
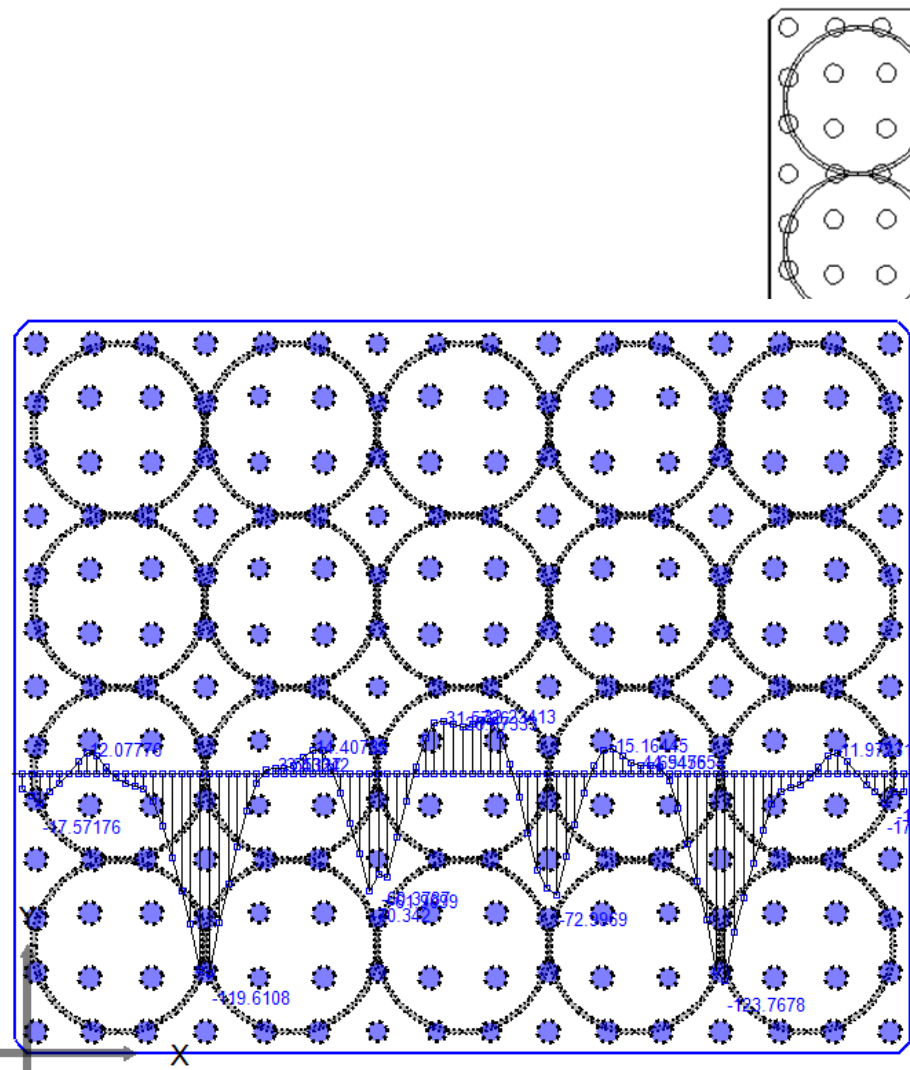
4. Foundation Package (FoundaPAK)



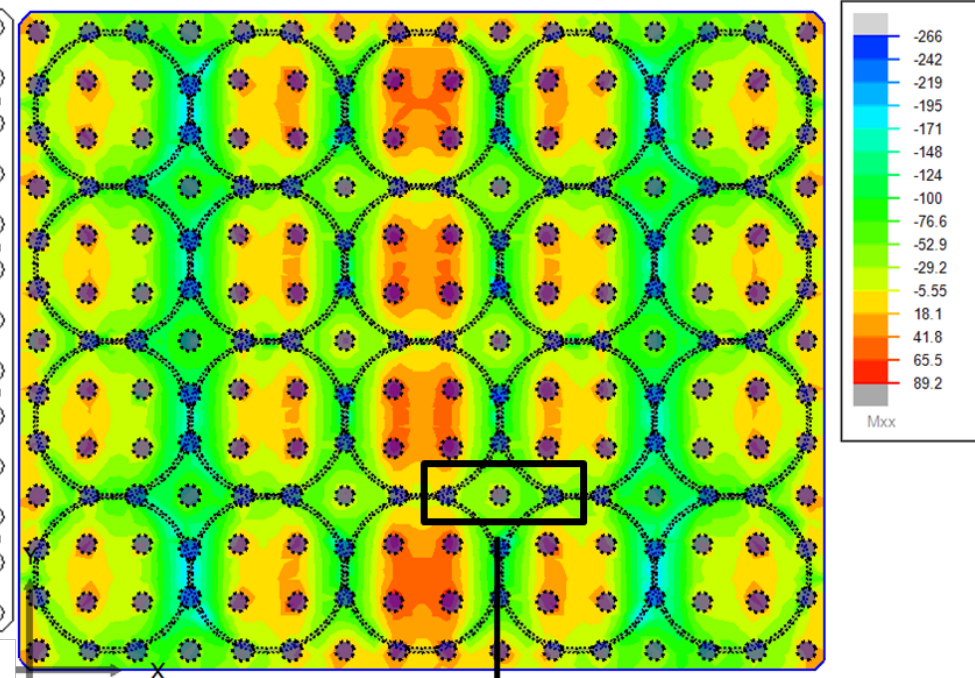
4. Foundation Package (Foundation)



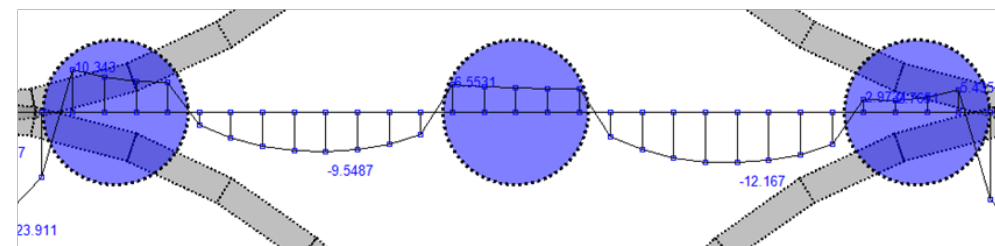
4. Foundation Package (Foundations)



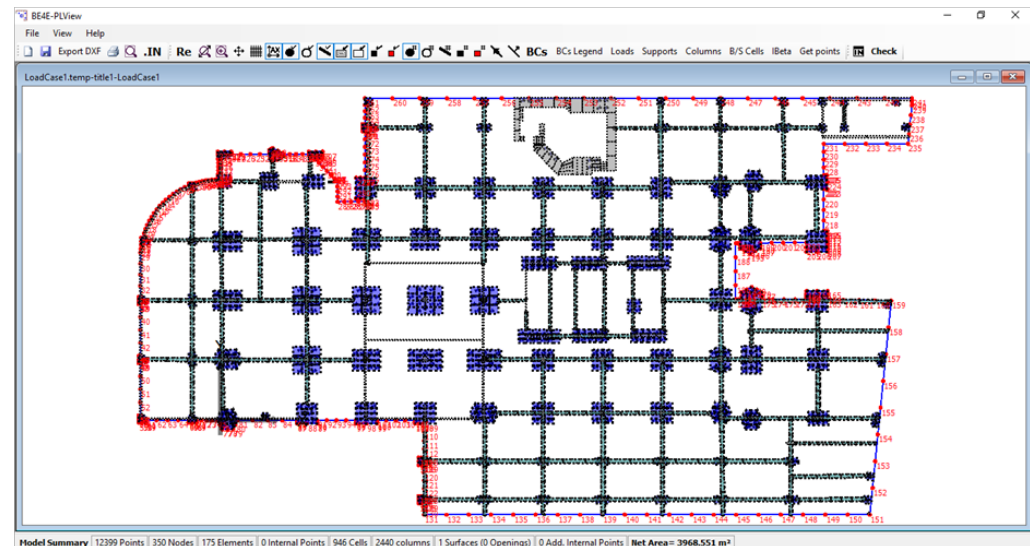
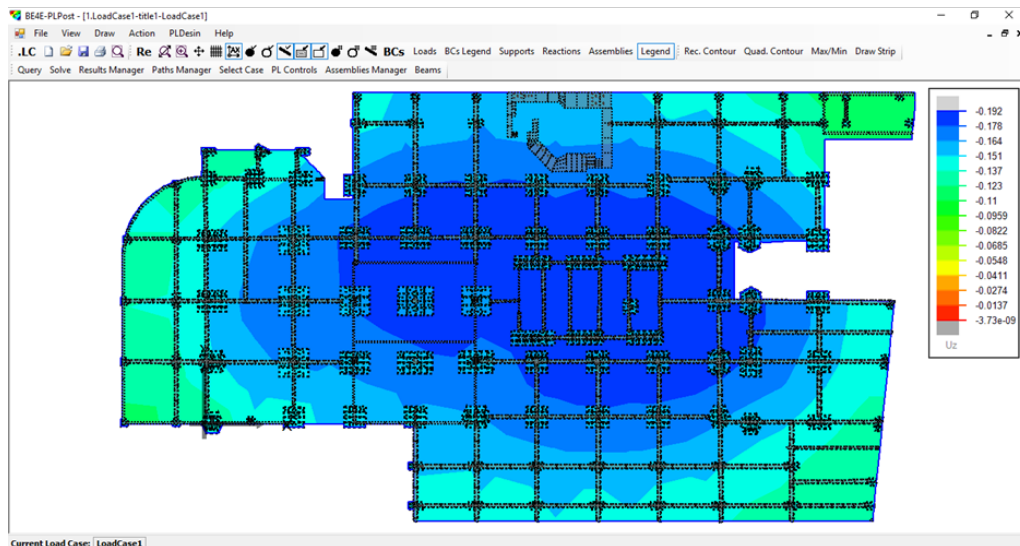
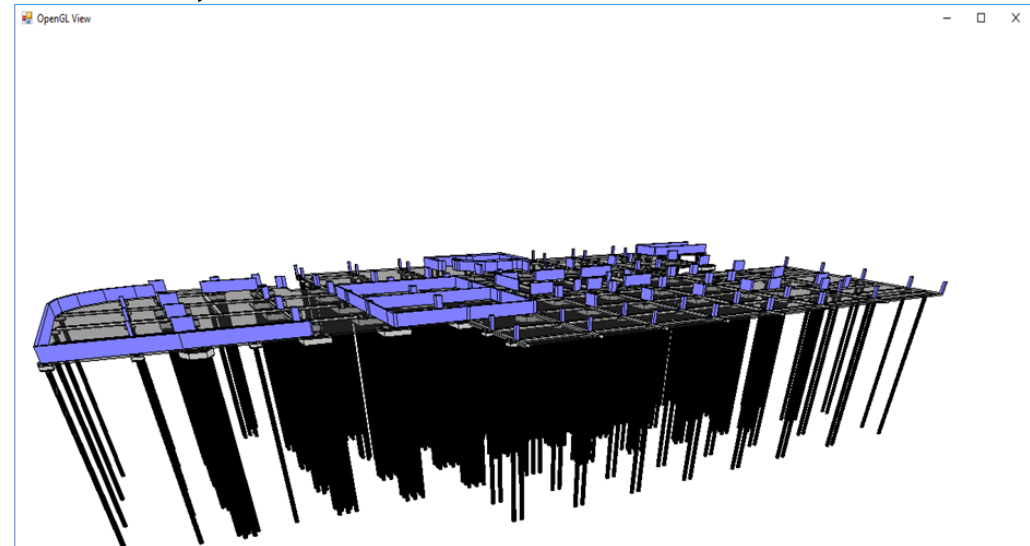
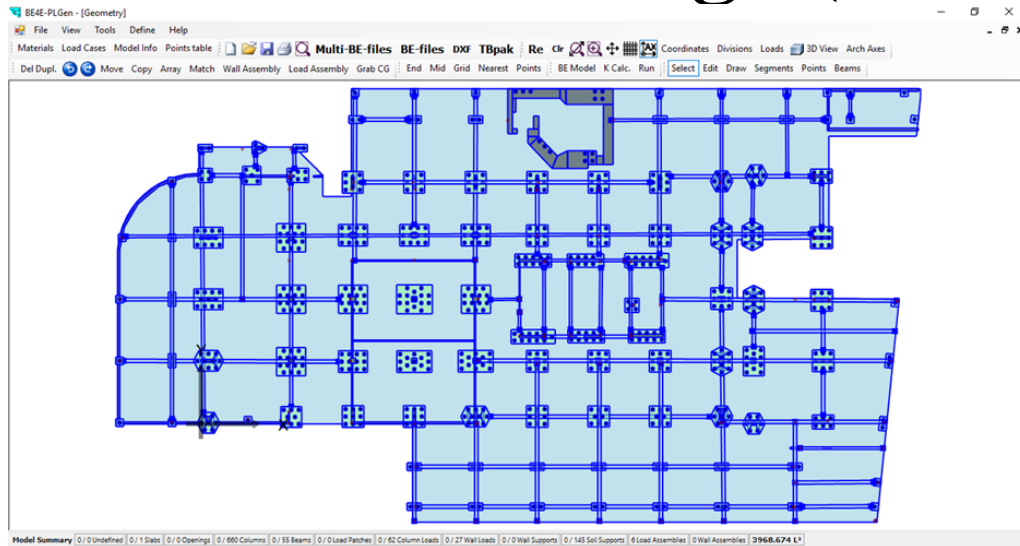
AutoCAD Drawing



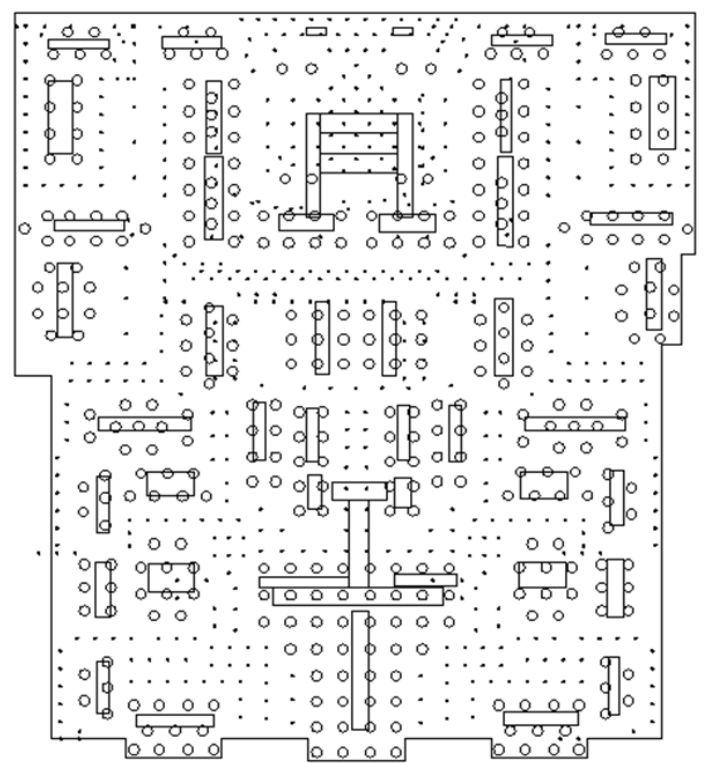
Bending Moment contour at x-direction



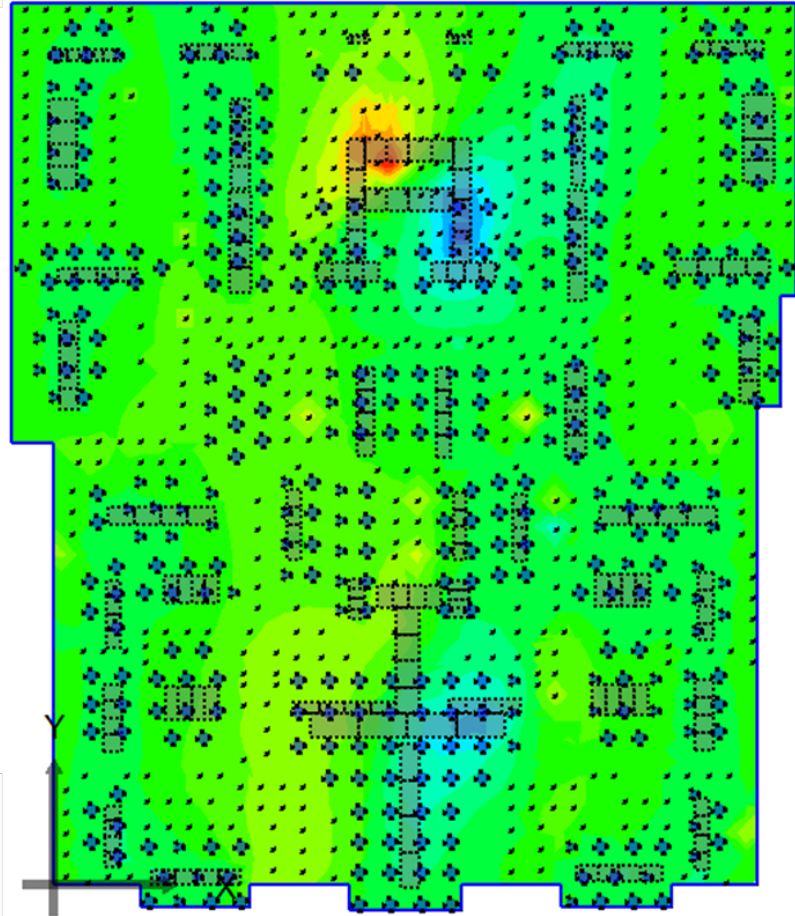
4. Foundation Package (Foundation)



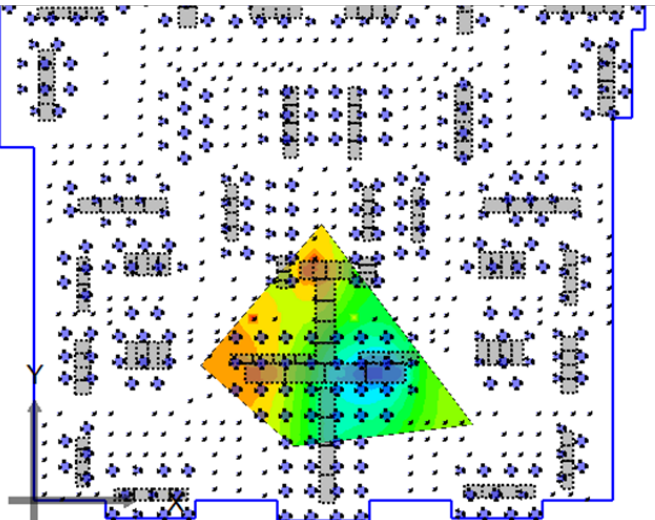
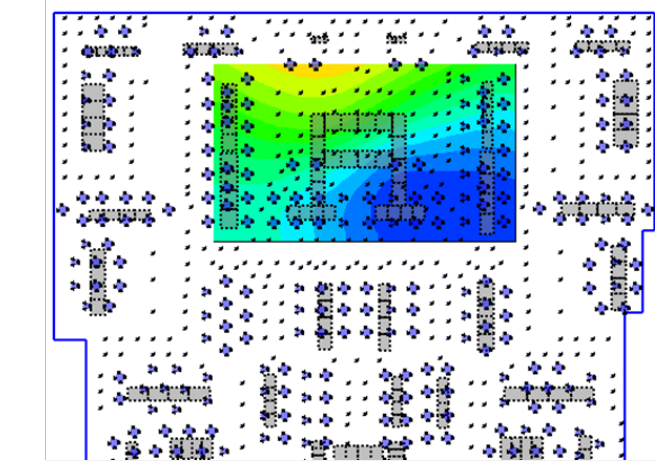
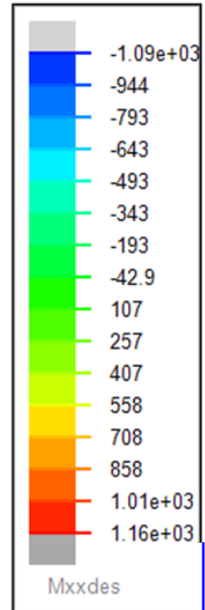
4. Foundation Package (Foundation K)



AutoCAD Drawing

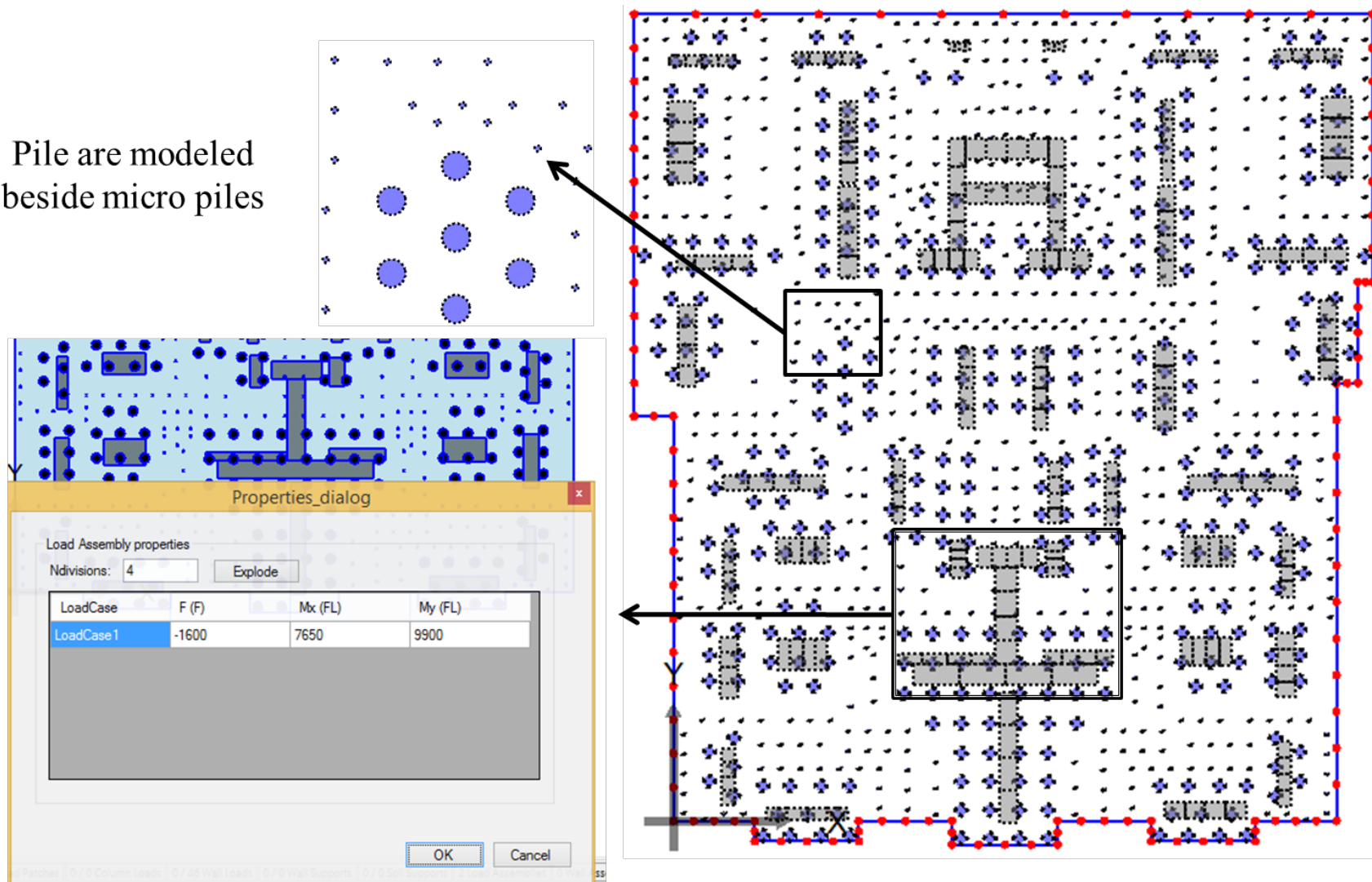


Bending Moment design contour at x-direction



4. Foundation Package (Foundation)

Pile are modeled beside micro piles



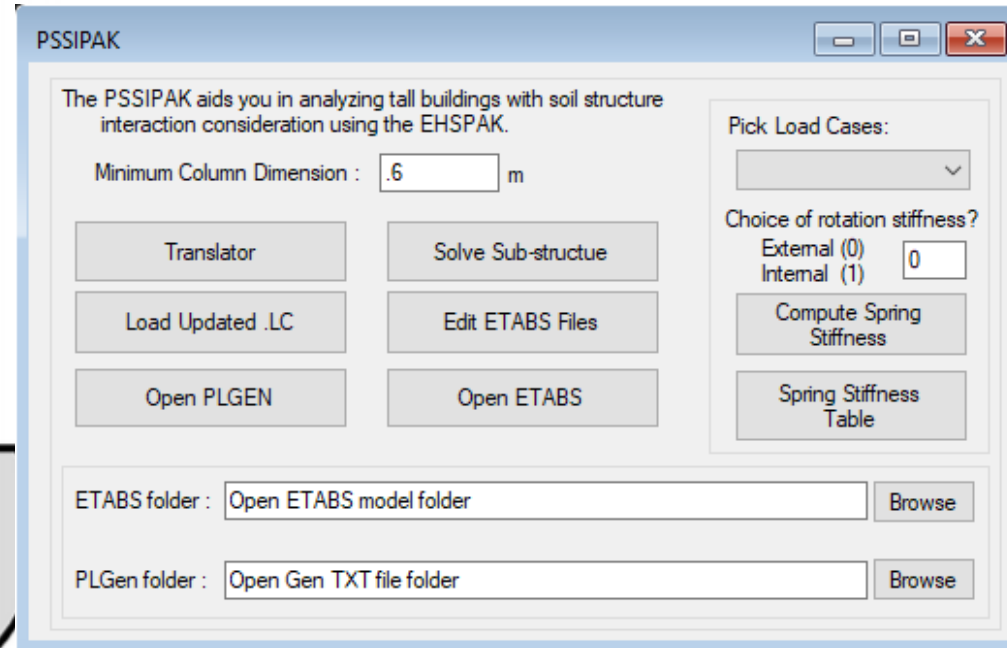
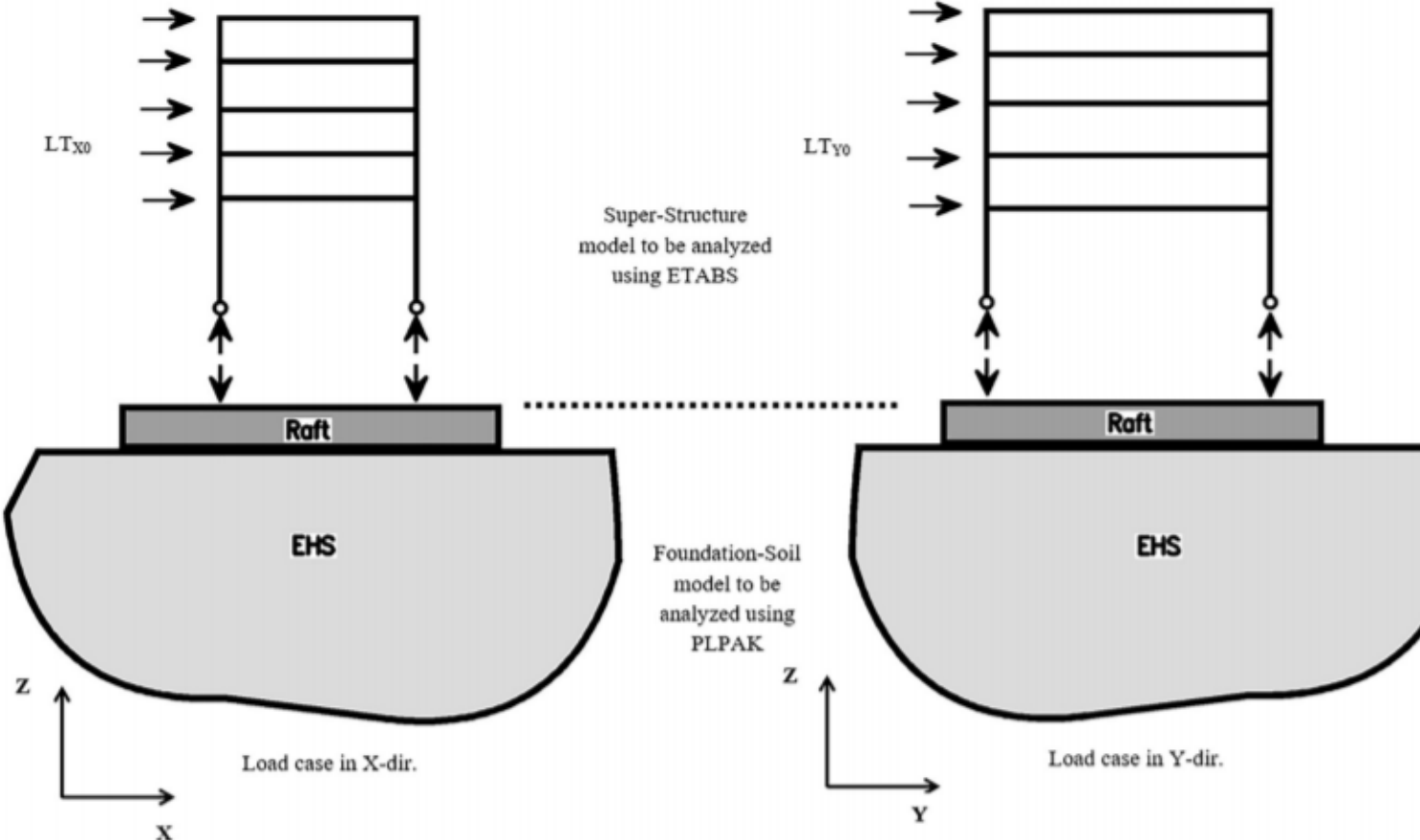
The screenshot displays a software interface for modeling a foundation package. The main view shows a grid of piles (represented by blue dots) and micro piles (represented by blue crosses) arranged in a rectangular pattern. A central structure is visible, consisting of a vertical column and horizontal beams. A properties dialog box is open, showing the following data:

LoadCase	F (F)	Mx (FL)	My (FL)
LoadCase 1	-1600	7650	9900

The dialog box also includes a table for 'Load Assembly properties' with 'Ndivisions' set to 4 and an 'Explode' button. The main view includes a coordinate system with X, Y, and Z axes and a grid of red dots representing the boundary of the foundation package.

4. Foundation Package (FoundPAK)

User can solve building taking into accounts soil-structure interaction by using ETABS-PLPAK iterative solution. Super structure modeled using ETABS software, on the other hand raft/piled raft modeled using FoundPAK.



CUFE-BE



www.be4e.com

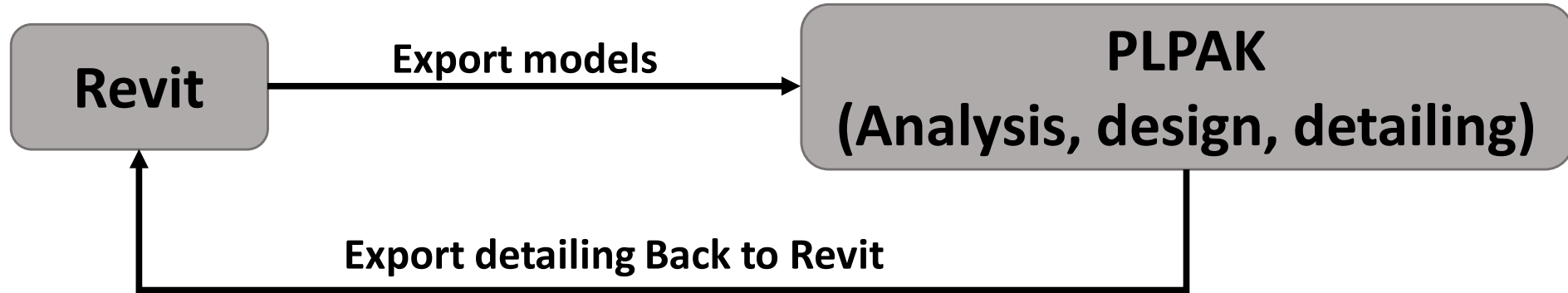


Cairo University

Table of contents

- 1. Introduction**
- 2. Basic package (PLPAK Basic)**
- 3. One floor package (BIM-PLPAK)**
- 4. Foundation Package (FoundPAK)**
- 5. Fixed base Package (FBPAK)**
6. Post-tension Package (PTPAK)
7. Dynamic package (DynPAK)
8. Overall building package (OBPAK)
9. 4D and 5D analysis
10. Conclusions

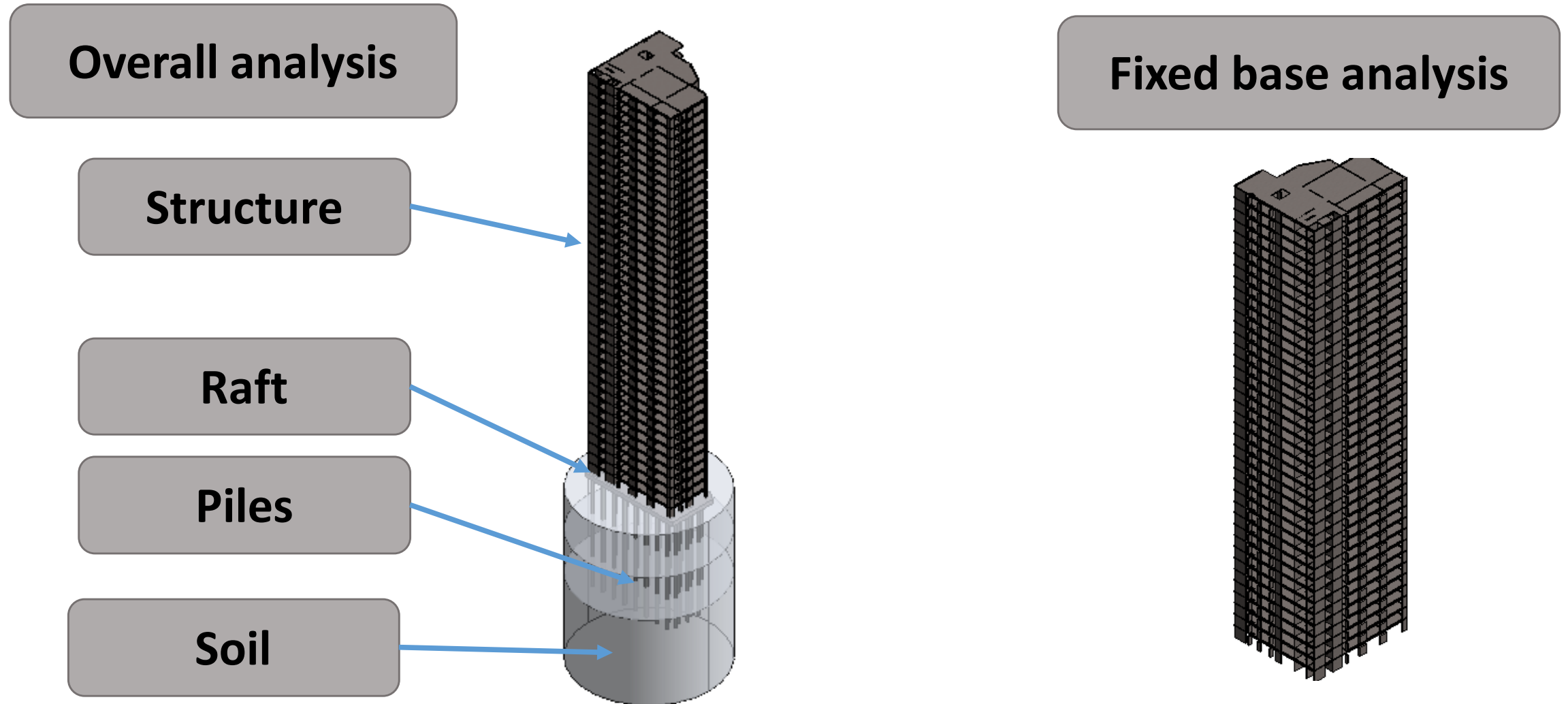
5. Fixed base Package (FBPAK)



- BIM centered
- Lean process
- Quantity takeoff

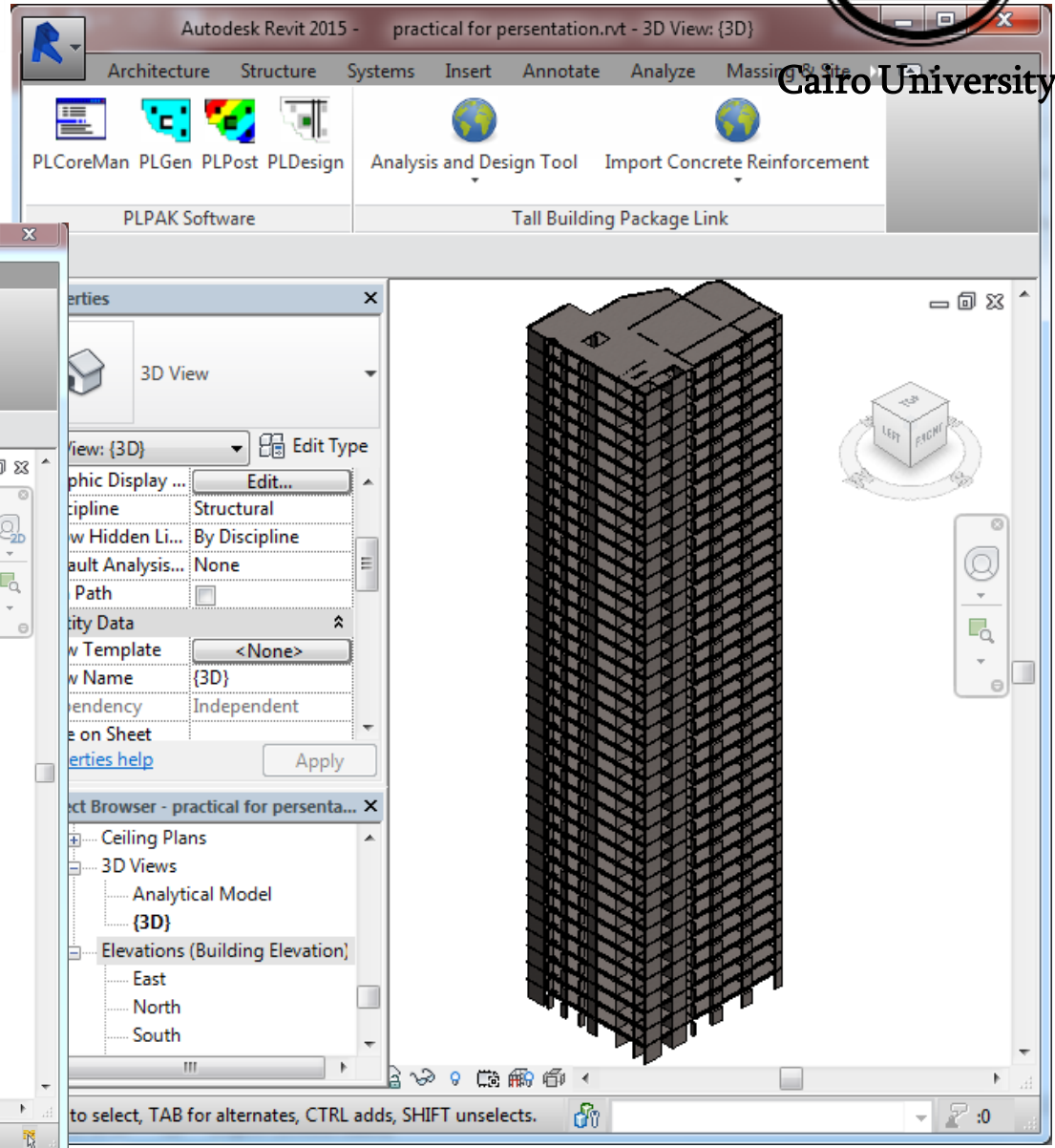
5. Fixed base Package (FBP)

www.be4e.com



5. Fixed base Package (FBPAK)

www.be4e.com



The image shows a screenshot of Autodesk Revit 2015. The main window displays a 3D view of a tall building model. The interface includes a ribbon with tabs for Architecture, Structure, Systems, Insert, Annotate, Analyze, and Massing & Site. Below the ribbon, there are toolbars for PLCoreMan, PLGen, PLPost, PLDesign, Analysis and Design Tool, and Import Concrete Reinforcement. The Properties panel on the left shows the 'Building Elevation' properties, including Elevation: East, Visibility/Graphics, Graphic Display, Hide at scales, Discipline: Structural, Show Hidden Li..., Color Scheme, Default Analysis, Reference Label, Sun Path, and Properties help. The Project Browser on the left shows the hierarchy: Ceiling Plans, 3D Views, Analytical Model (3D), Elevations (Building Elevation), East, North, and South. The Tall Building Tool dialog box is open in the center, displaying the following text:

Tall Building Tool

The following is the process for analysis and design of a tall building on fixed supports. Follow the steps below sequentially.

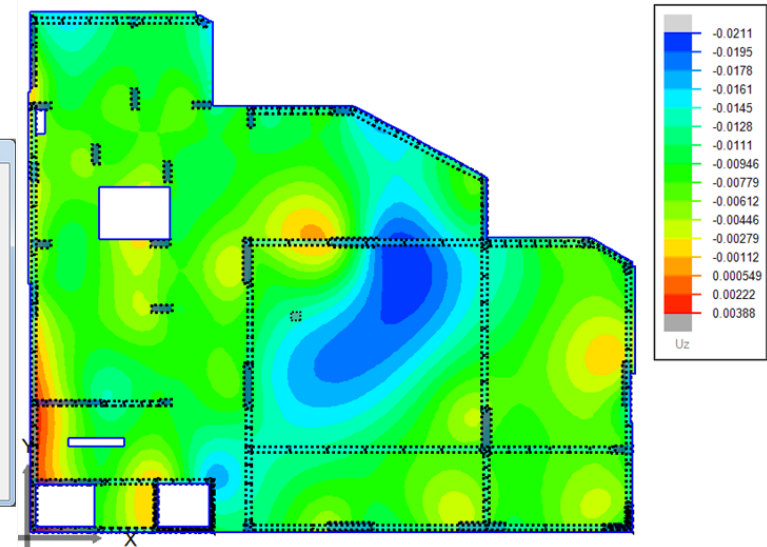
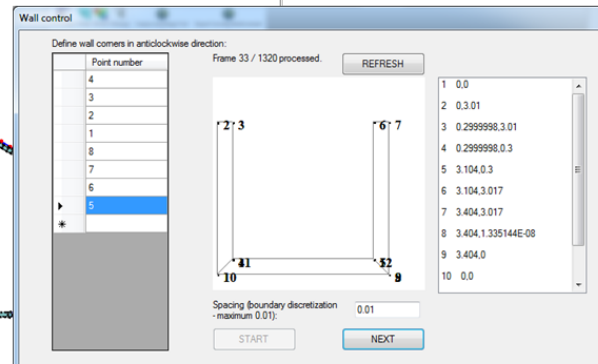
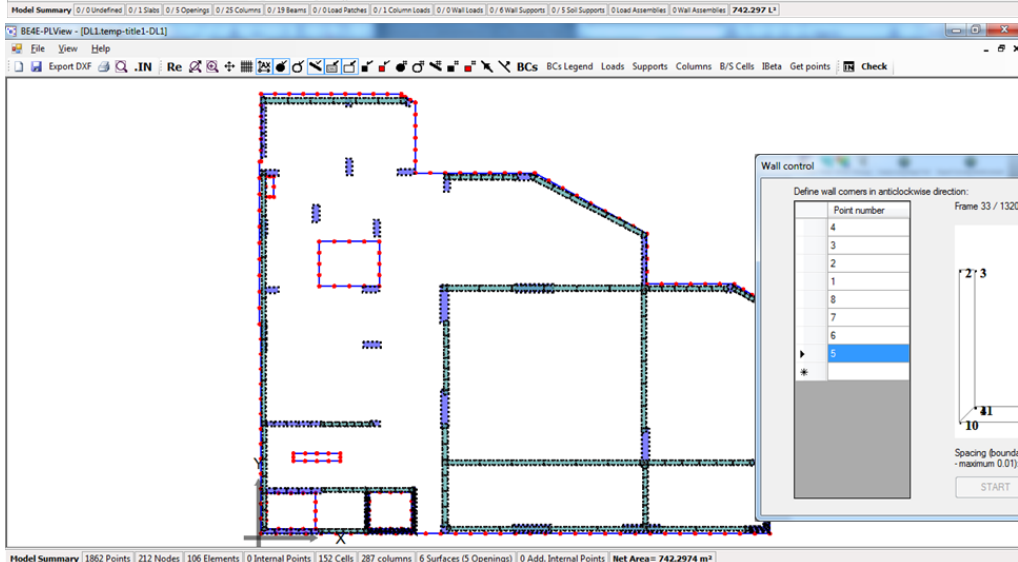
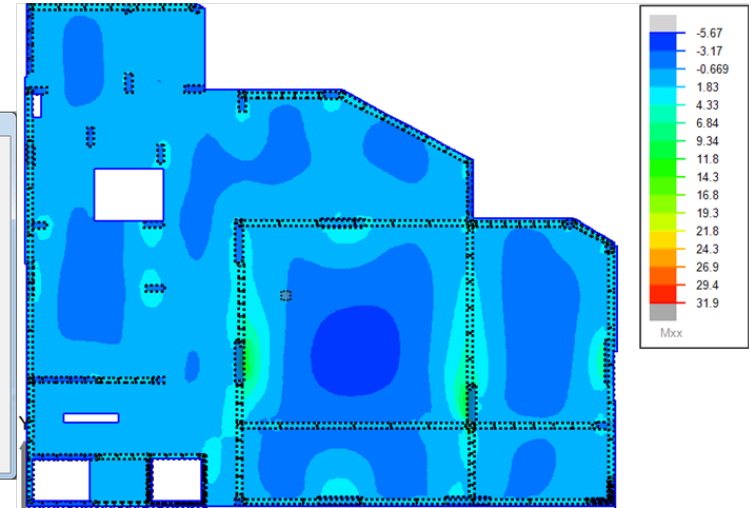
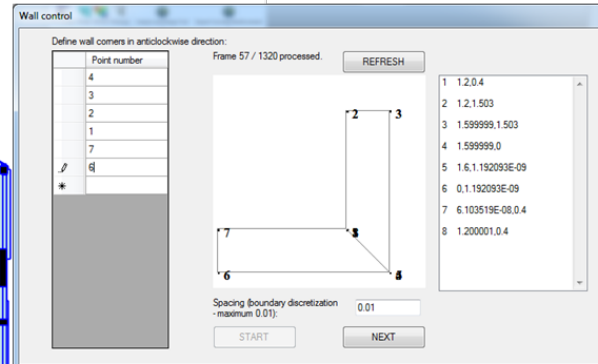
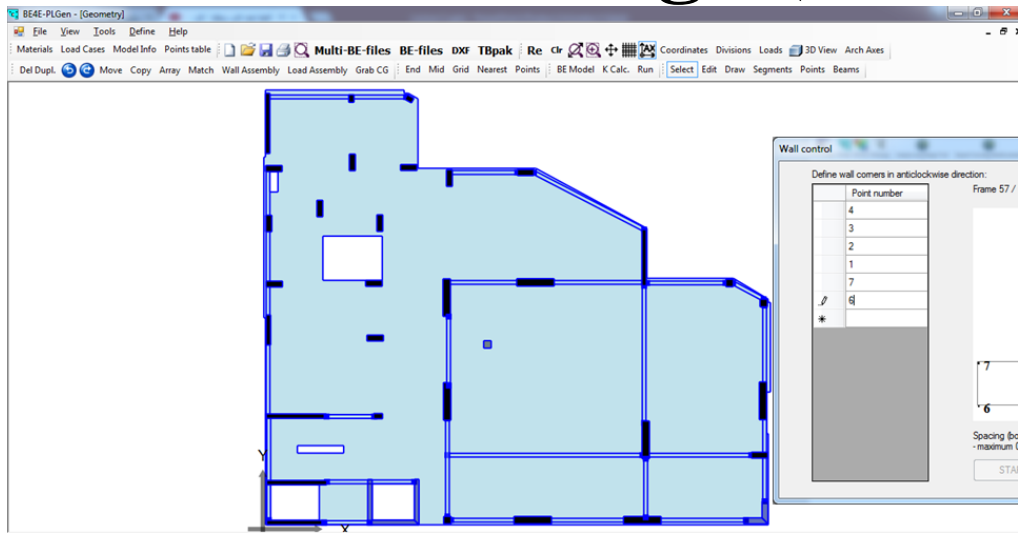
- 1- Export text files for PLPAK.
- 2- Convert and create all BEFILEs.

You may choose either :
step 3 (solve on fixed base)
or step 4 (extract stiffness only).
- 3- Run the TBPAK to solve your structure.
 Solve on GPU (check to solve on GPU)
- 4- Postprocess with either PLPOST or PLDesign.
 List of Floors: (Click to view in either PLPOST or PLDesign)
Floor_ID_217457;Floor_ID_217457
- 5- View in OpenGL viewer and design vertical elements.
- 6- Show reinforcement detailing.

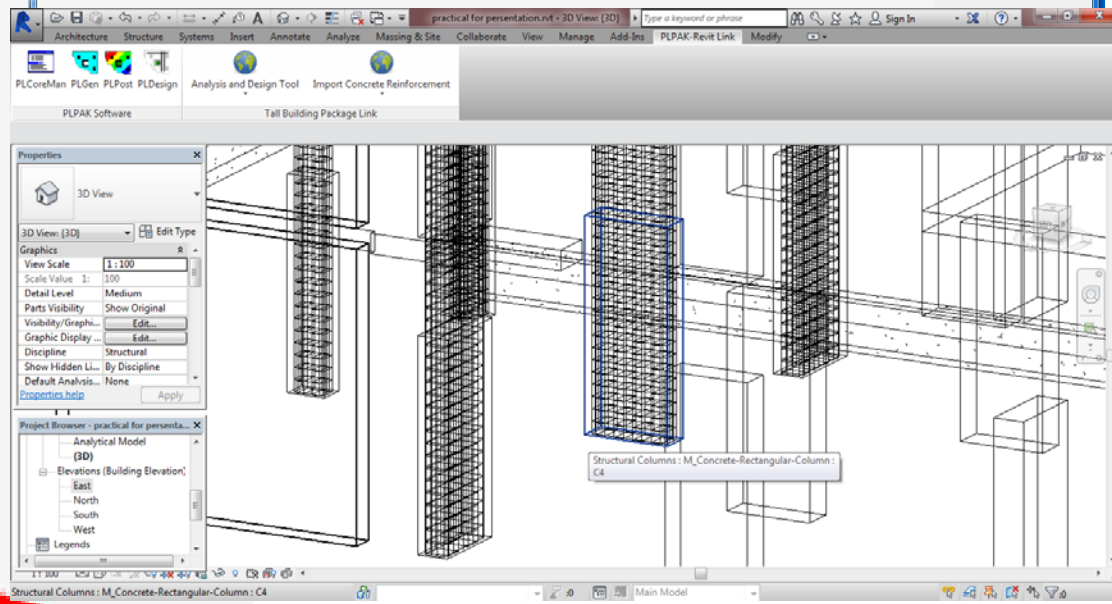
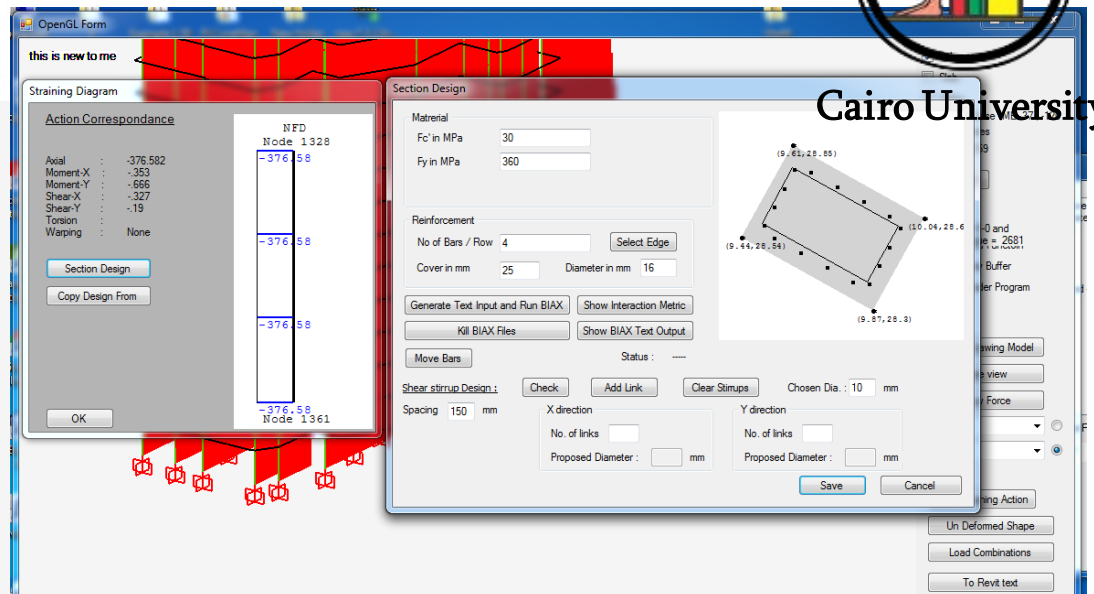
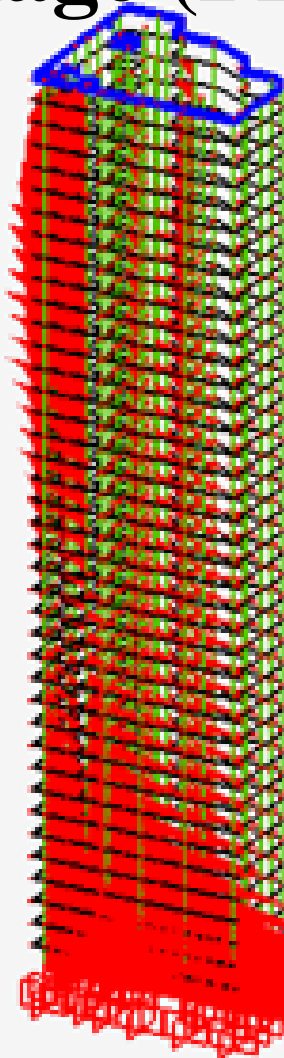
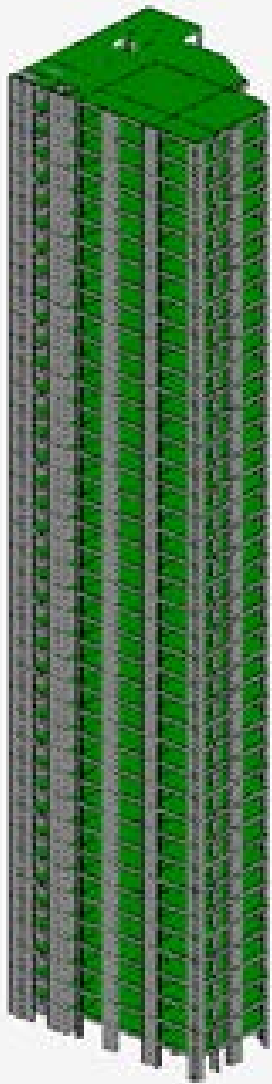
Log of actions:
Structural components and loading have been exported.
Boundary element files have been created for each floor.
Step 3 chosen, solving on fixed base.
Translator has modified Revit files.
BEAMMODE has been modified.
RUNCONTROL has been modified.
Solving, please wait ...
TBPAK has run successfully and solved on fixed base.

The Properties panel on the right shows the '3D View' properties, including Edit Type, Discipline: Structural, Show Hidden Li..., Default Analysis, Path, City Data, View Template, View Name (3D), Dependency: Independent, and Properties help. The Project Browser on the right shows the hierarchy: Ceiling Plans, 3D Views, Analytical Model (3D), Elevations (Building Elevation), East, North, and South.

5. Fixed base Package (FBP)



5. Fixed base Package (FBPAK)



CUFE-BE



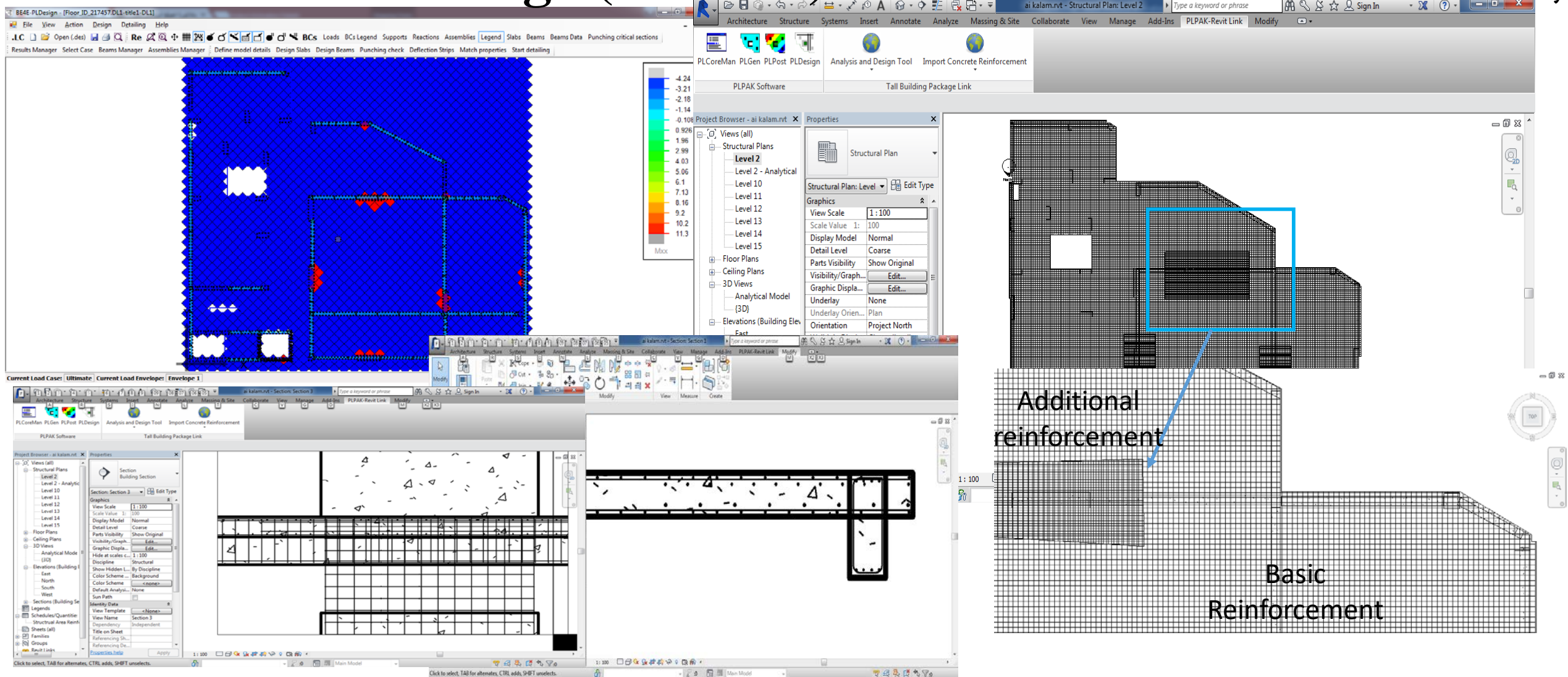
Cairo University

5. Fixed base Package (FBPAK)

www.be4e.com



5. Fixed base Package (FBPAK)



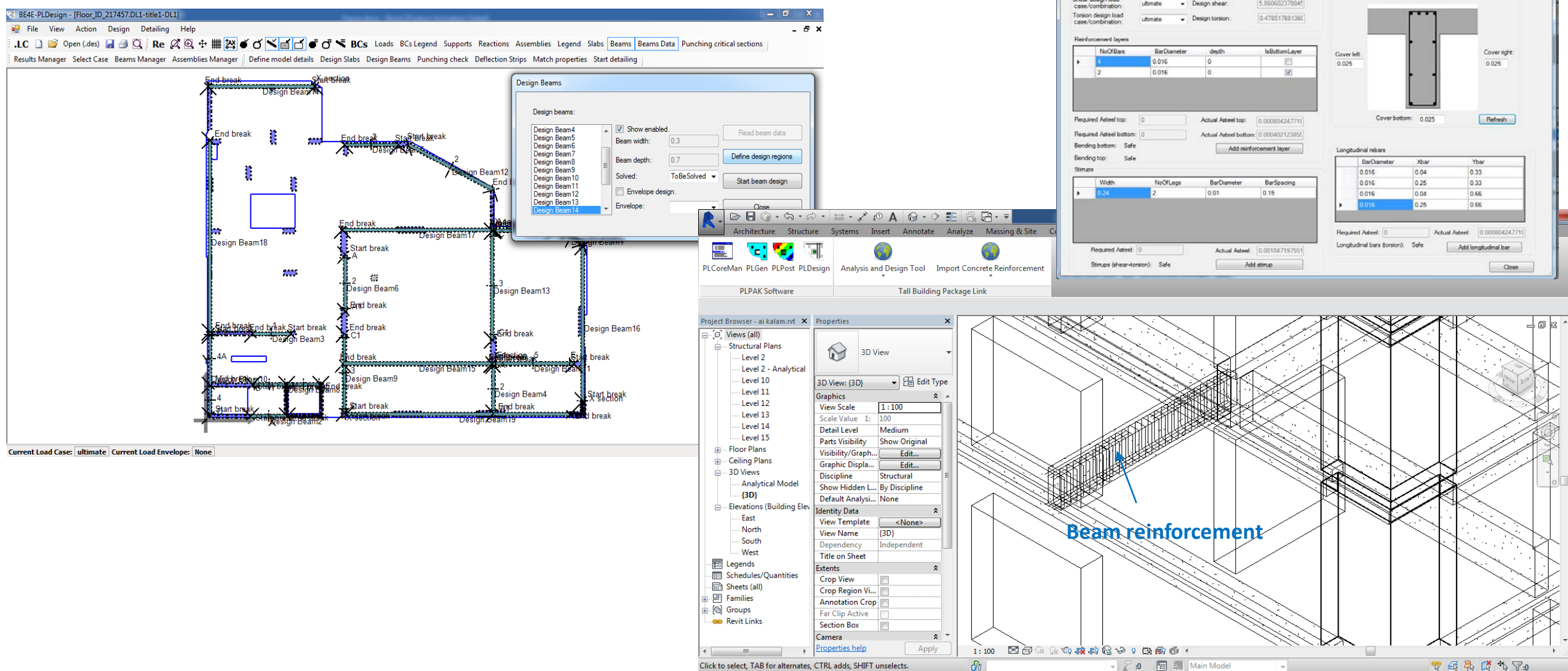
The screenshot displays the PLPAK software interface for structural analysis and design. The main window shows a 3D model of a building structure with a color-coded stress distribution. A vertical color scale on the right indicates stress values from -4.24 (blue) to 11.3 (red). The interface includes a Project Browser on the left, a Properties panel on the right, and a central workspace showing a 3D model of the structure. A blue box highlights a specific area of the structure, which is further detailed in a 3D view below. This 3D view compares 'Basic Reinforcement' (a standard grid) with 'Additional reinforcement' (a denser grid). The software interface also shows various toolbars, a menu bar, and a status bar.

Additional reinforcement

Basic Reinforcement

5. Fixed base Package (FBPAK)

www.be4e.com



The screenshot displays the PLPAK software interface for structural design. The main window shows a 2D plan view of a building floor with various design beams (Design Beam 3 to Design Beam 18) and their connections. A 'Design Beams' dialog box is open, showing parameters for Design Beam 14: Beam width: 0.3, Beam depth: 0.7, Solved: ToBeSolved, and Envelope design: . A 'Define reinforcement details' dialog box is also open for Design Beam 11, showing reinforcement layers and required/actual steel values. The reinforcement layers table is as follows:

NoOfBars	BarDiameter	depth	IsBottomLayer
4	0.016	0	<input type="checkbox"/>
2	0.016	0	<input checked="" type="checkbox"/>

The required and actual steel values are: Required Asteel top: 0, Actual Asteel top: 0.000804247715; Required Asteel bottom: 0, Actual Asteel bottom: 0.000402123852. The 'Bending top' and 'Bending bottom' are set to 'Safe'. The 'Stumps' table is also visible:

Width	NoOfLegs	BarDiameter	BarSpacing
0.24	2	0.01	0.15

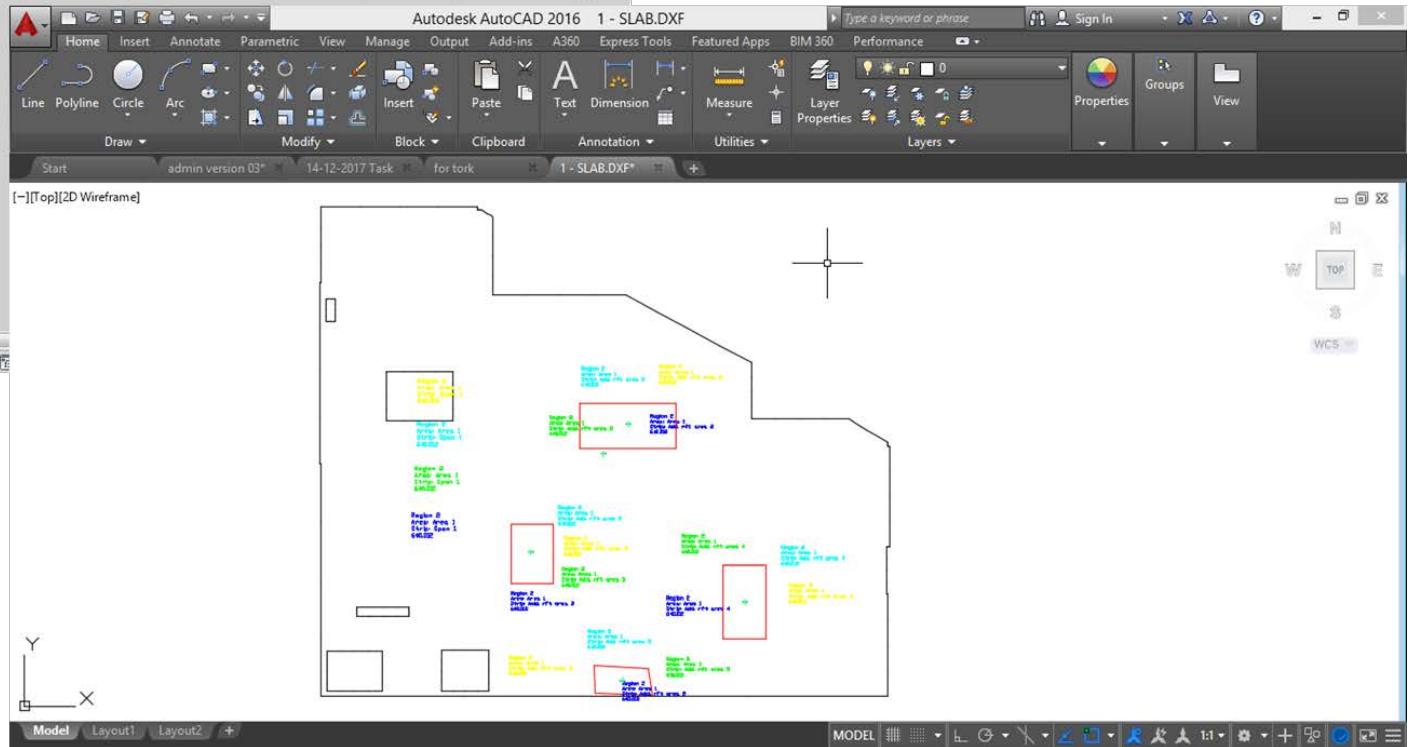
The 3D View window shows a perspective view of the beam reinforcement, with a blue arrow pointing to the reinforcement bars and the text 'Beam reinforcement'. The Project Browser on the left shows the current load case as 'ultimate' and the current load envelope as 'None'. The Properties window on the right shows the 3D View settings, including View Scale: 1:100 and Scale Value: 1:100.

5. Fixed base Package (FBP)

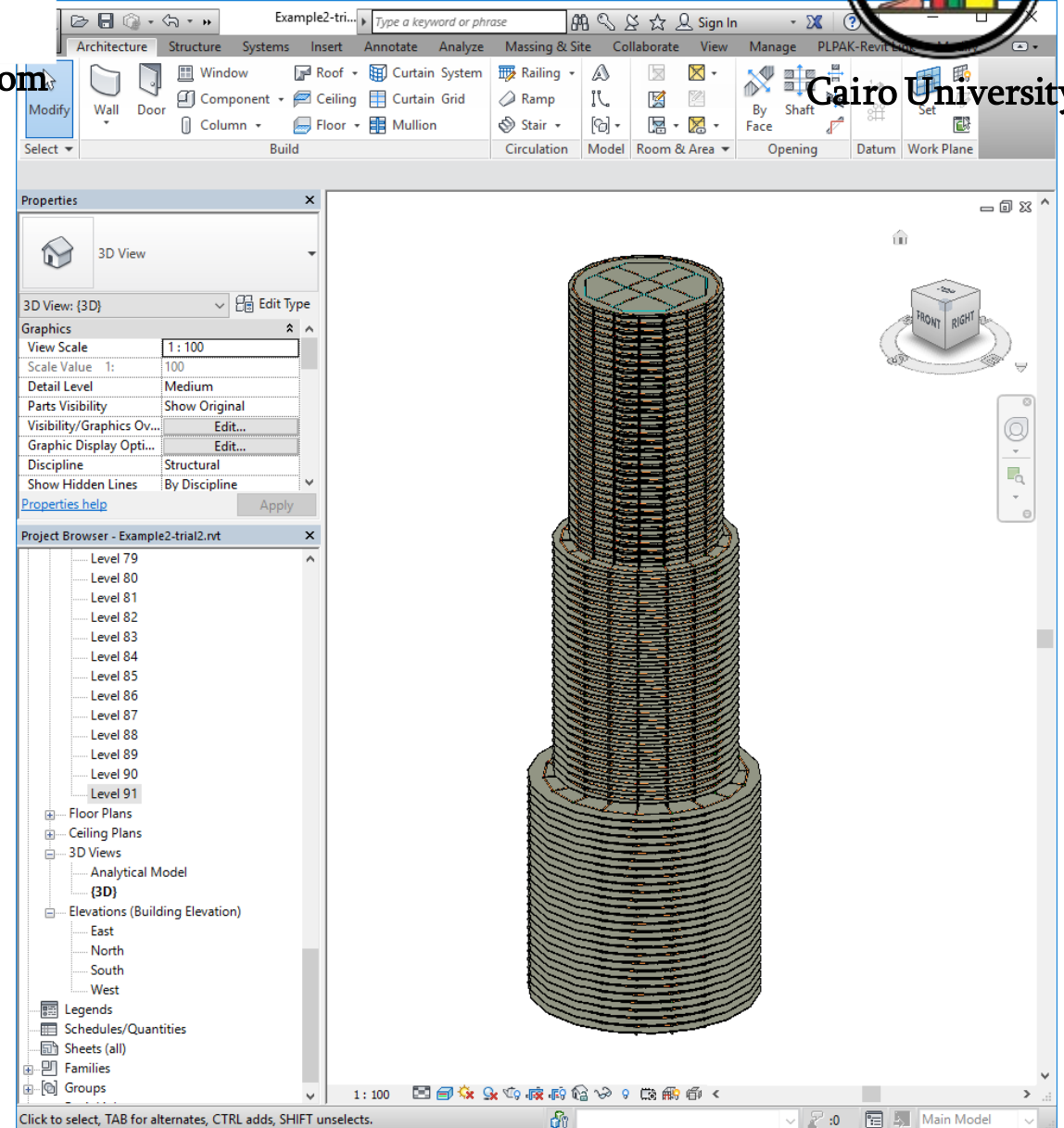
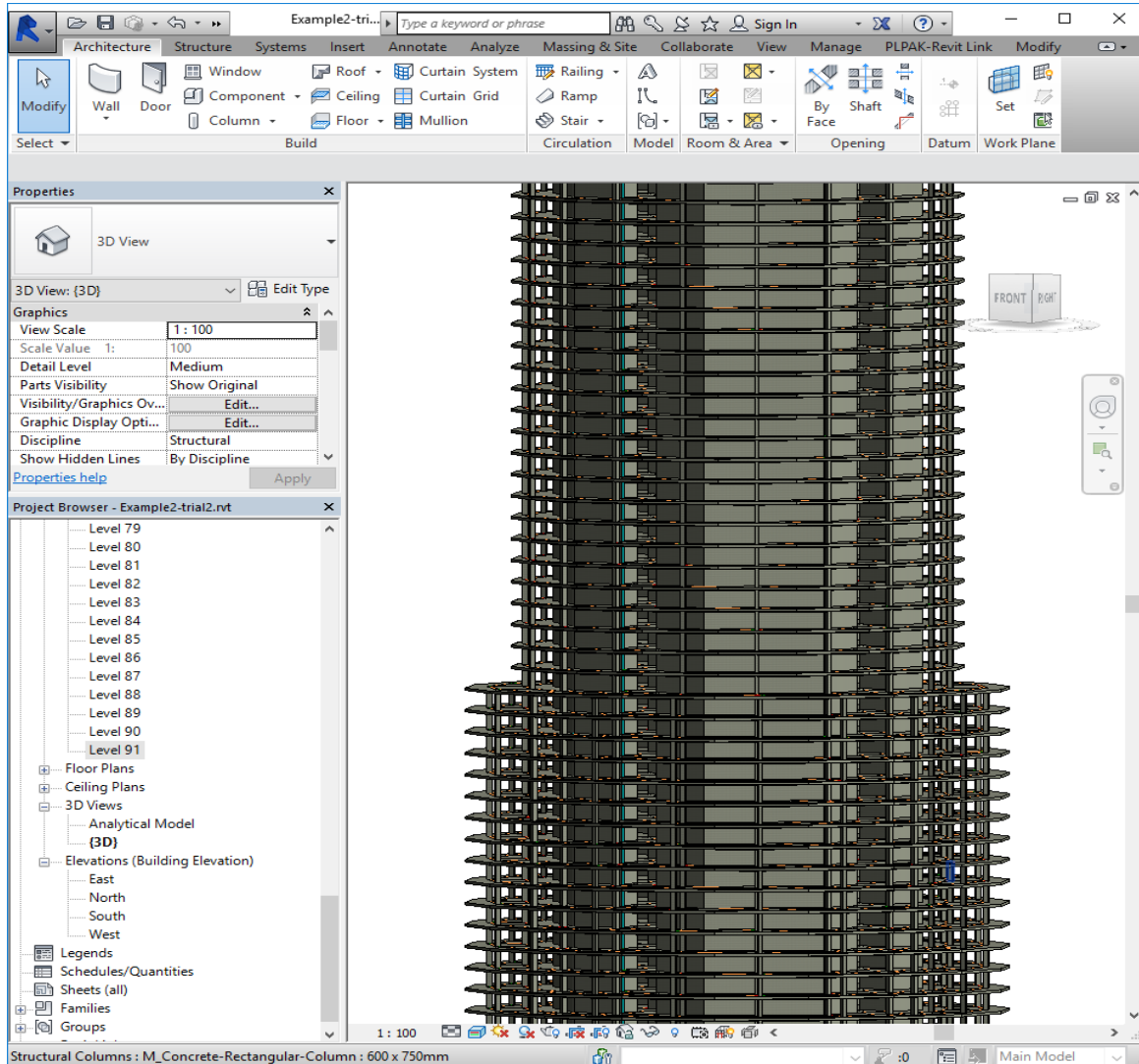
Autodesk Revit - Practical 1.rvt - Schedule: Structural Area Reinforcement

A	B	C	D	E	F	G	H
Bottom/Interior Major	Bottom/Interior Major (Brief)	Bottom/Interior Major	Bottom/Interior Major Spacing	Bottom/Interior Min	Bottom/Interior Min	Reinforcement Volume	Top/Exterior
Structural Rebar 1 @ 167 mm (B)	Structural Rebar 1 (B)	174	167 mm	202	167 mm	201454.31 cm ³	✓
Structural Rebar 1 @ 250 mm (B)	Structural Rebar 1 (B)	21	250 mm	10	300 mm	23184.95 cm ³	✓
Structural Rebar 1 @ 250 mm (B)	Structural Rebar 1 (B)	24	250 mm	2	300 mm	27633.45 cm ³	✓
Structural Rebar 1 @ 200 mm (B)	Structural Rebar 1 (B)	21	200 mm	2	300 mm	65370.26 cm ³	✓
Structural Rebar 1 @ 250 mm (B)	Structural Rebar 1 (B)	7	250 mm	2	300 mm	11862.65 cm ³	✓

Total Slab reinforcement volume = 2142595.62 cm³

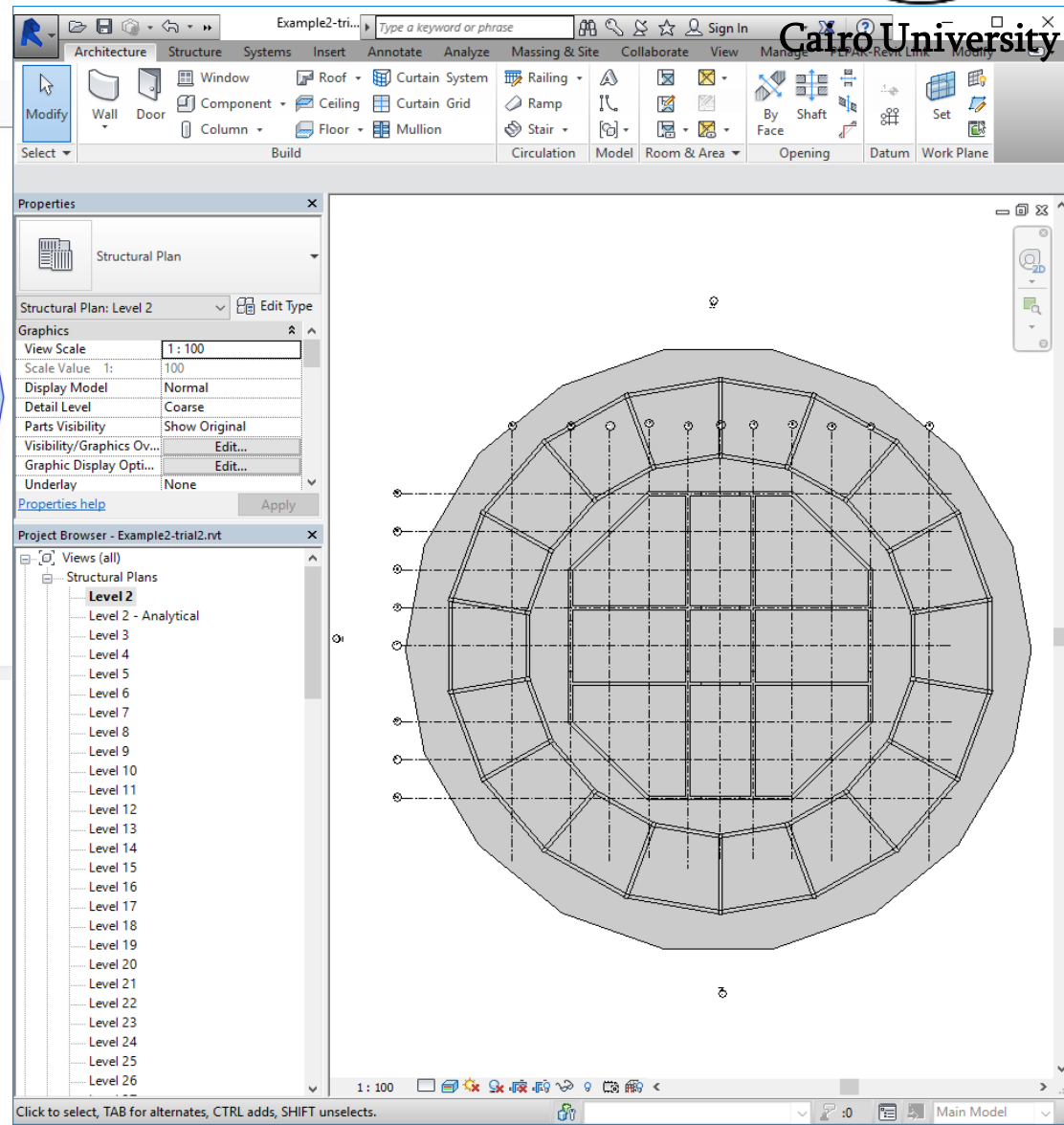
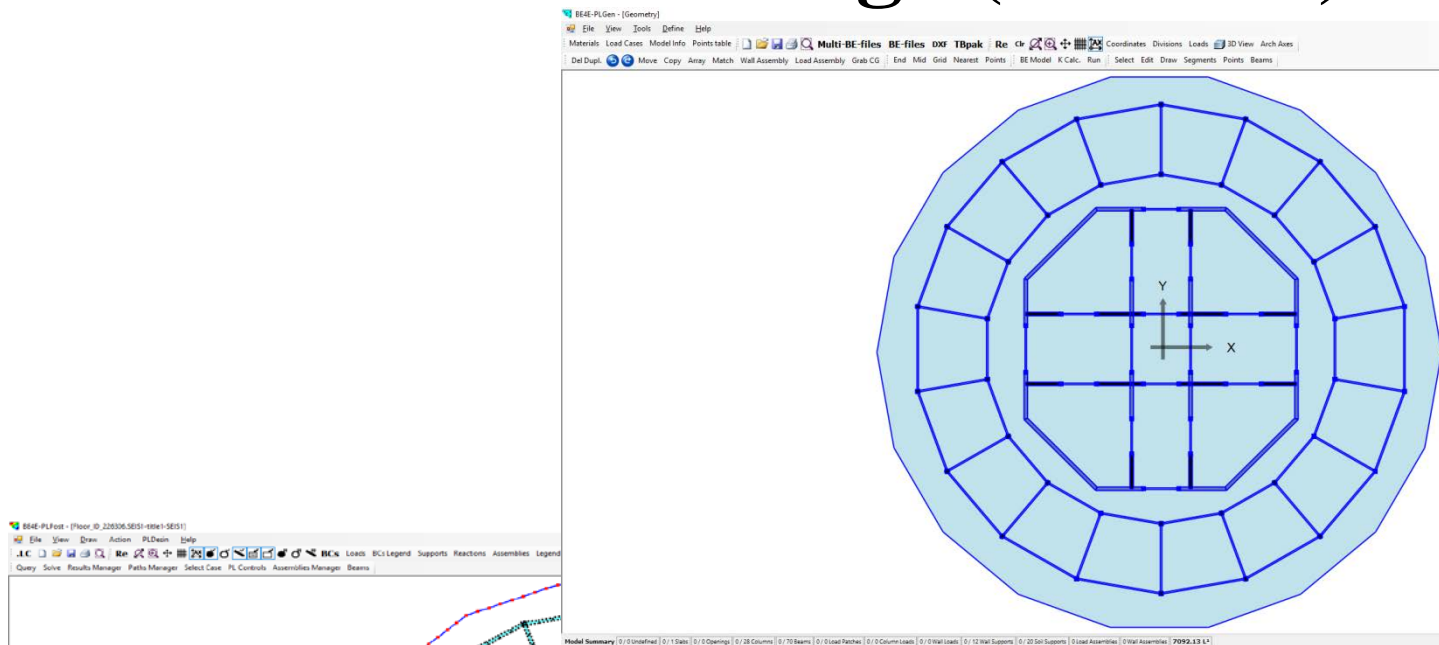


5. Fixed base Package (FBPAK)

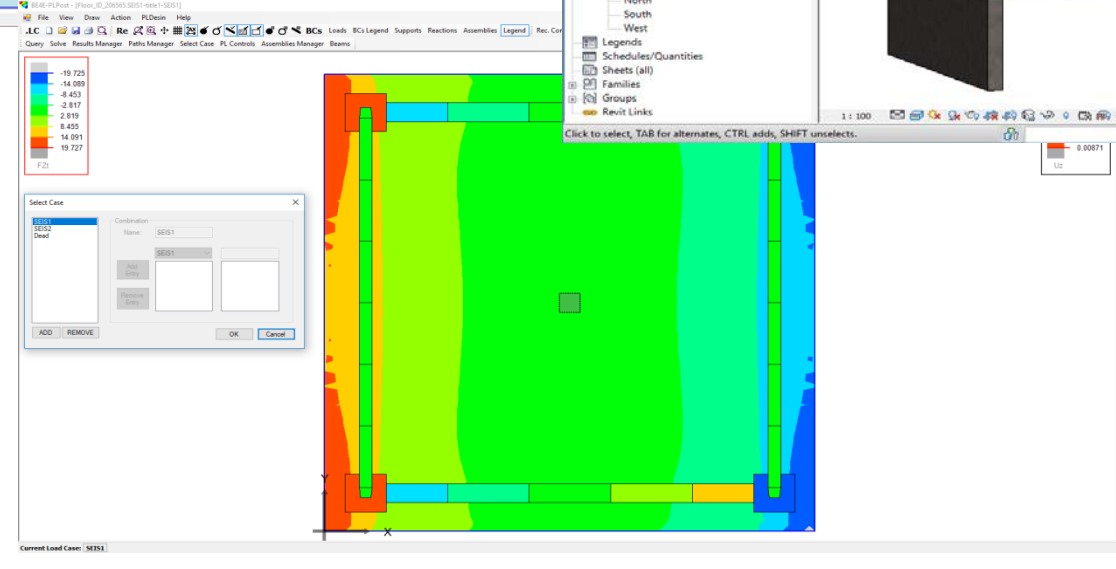
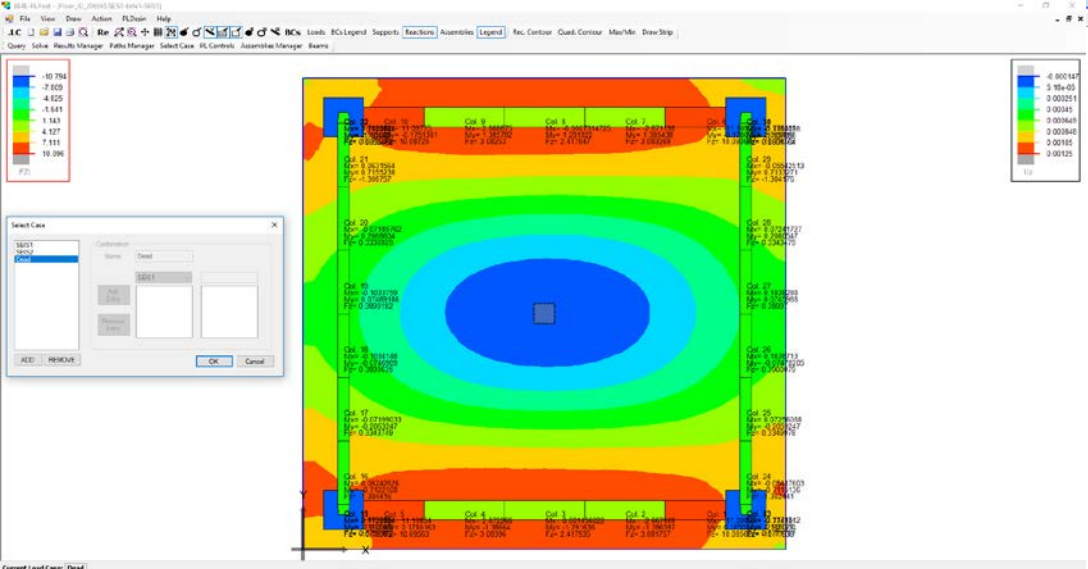
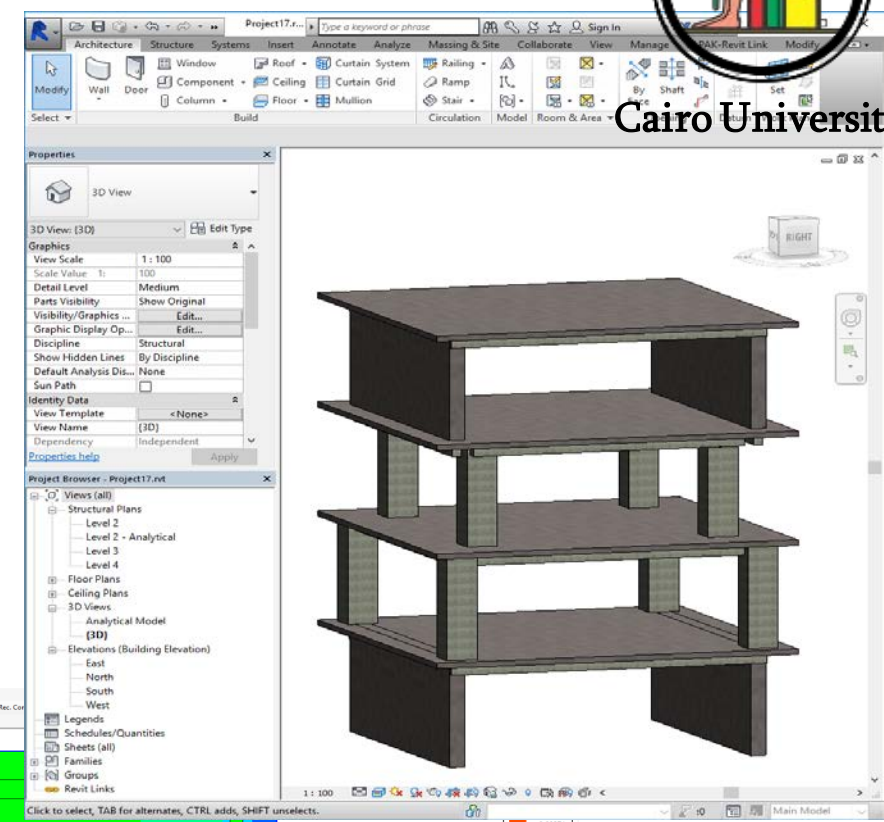
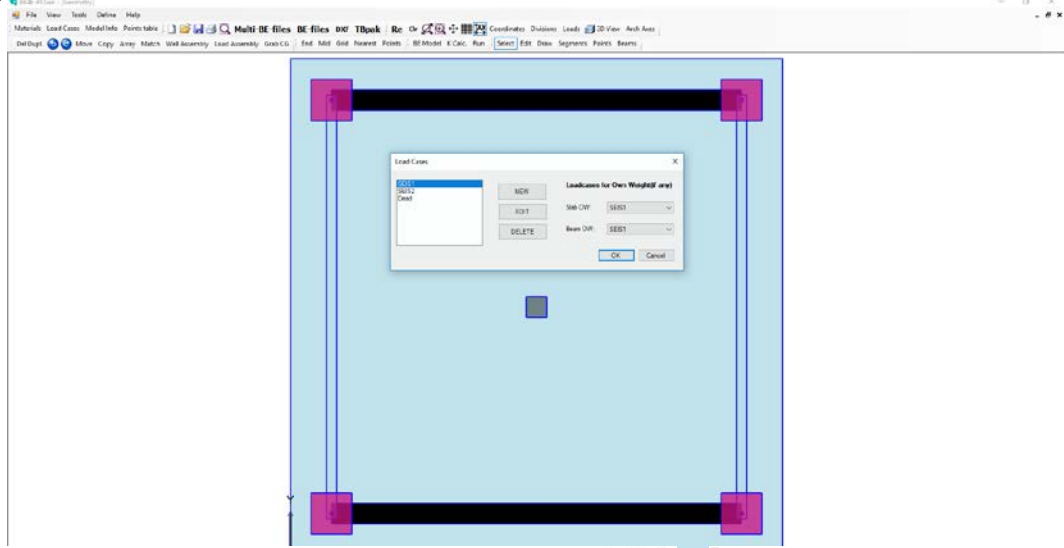


5. Fixed base Package (FBP)

www.be4e.com

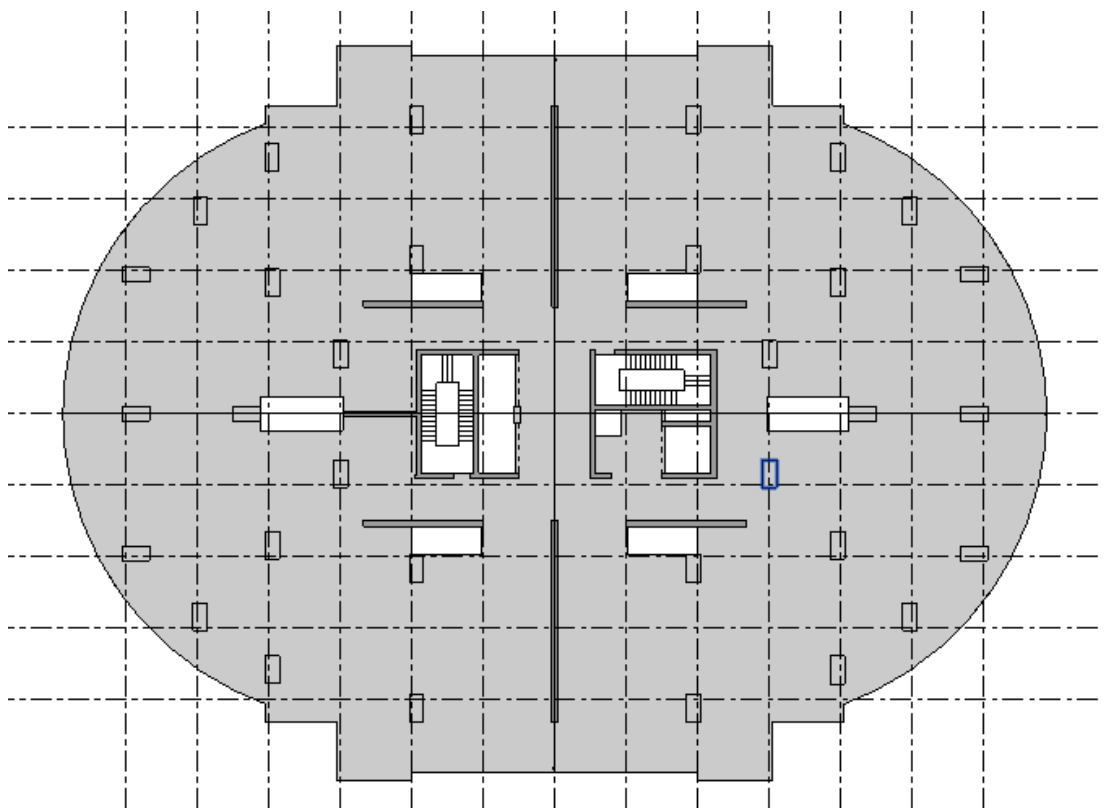


5. Fixed base Package (FBP)

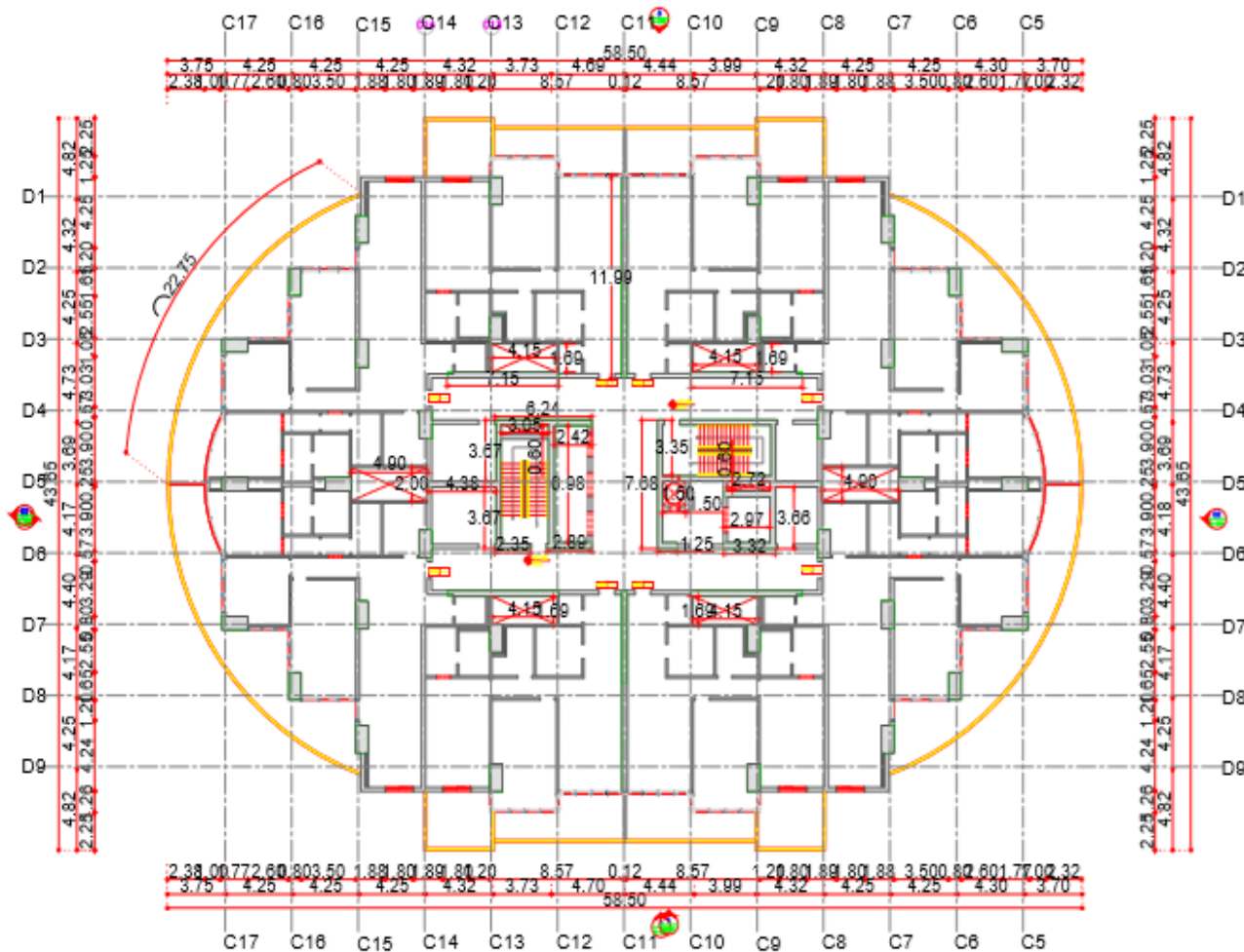


5. Fixed base Package (FBP)

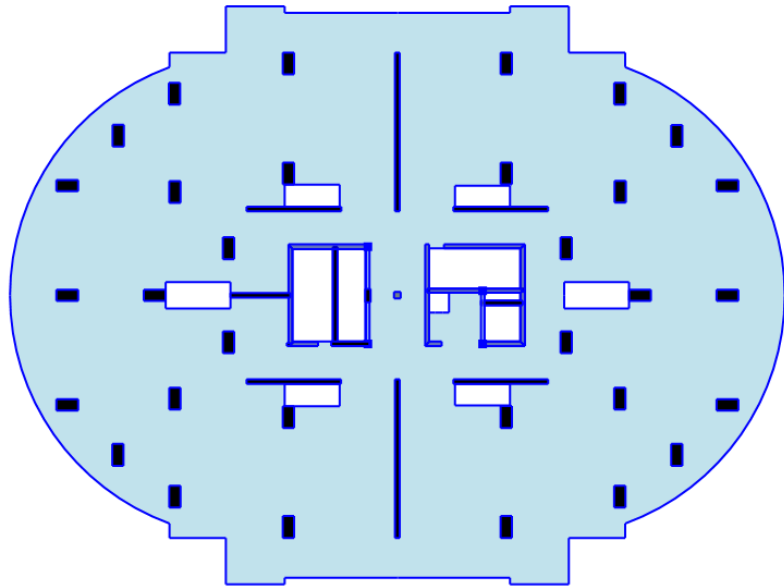
Cairo University



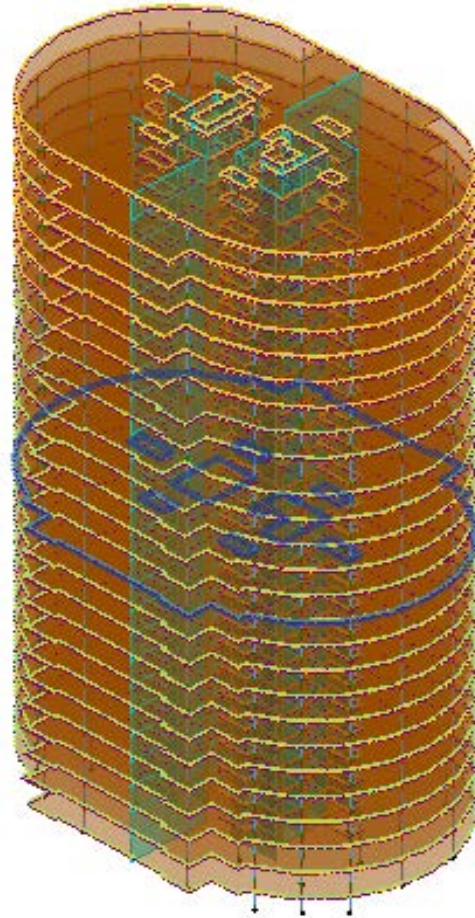
Plan view in Revit model



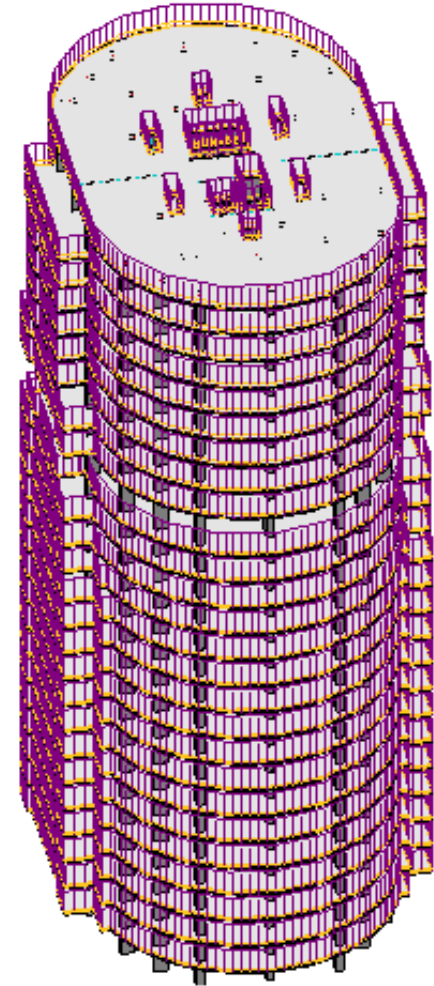
5. Fixed base Package (FBP)



PLGen model exported from Revit

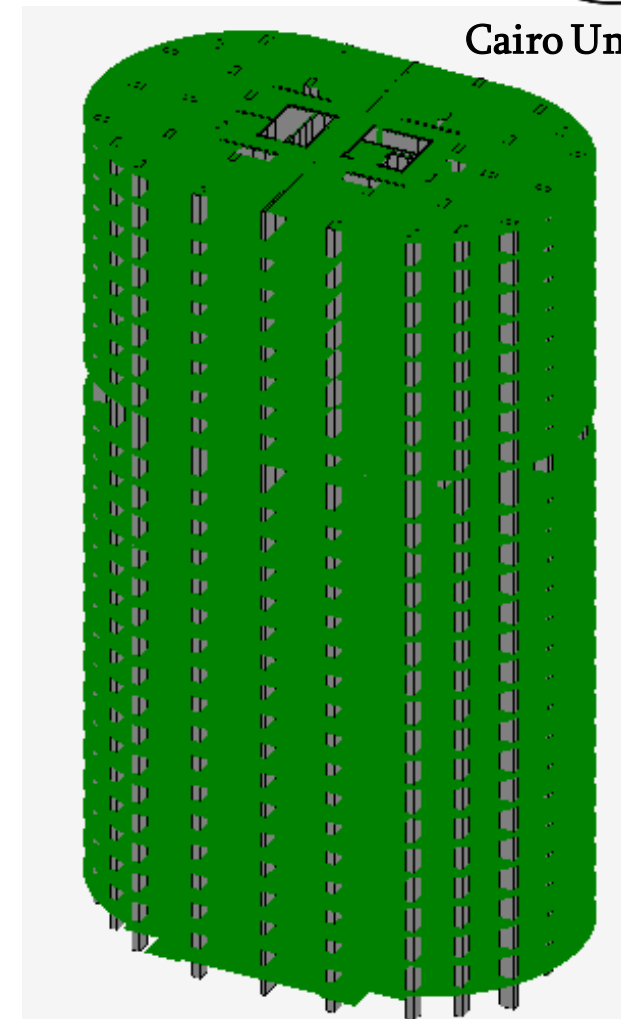
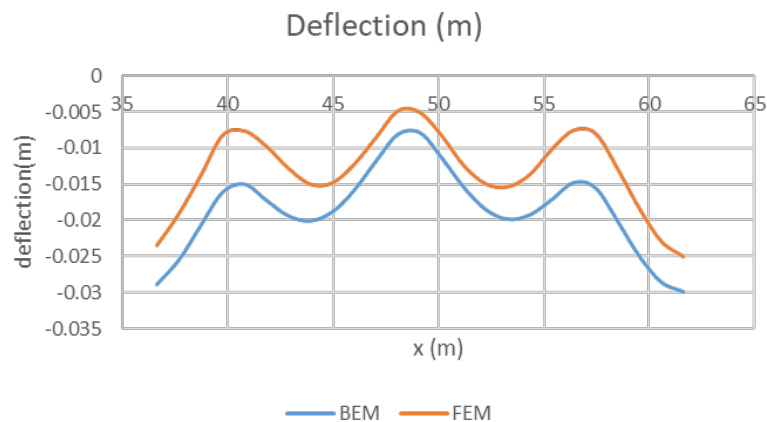
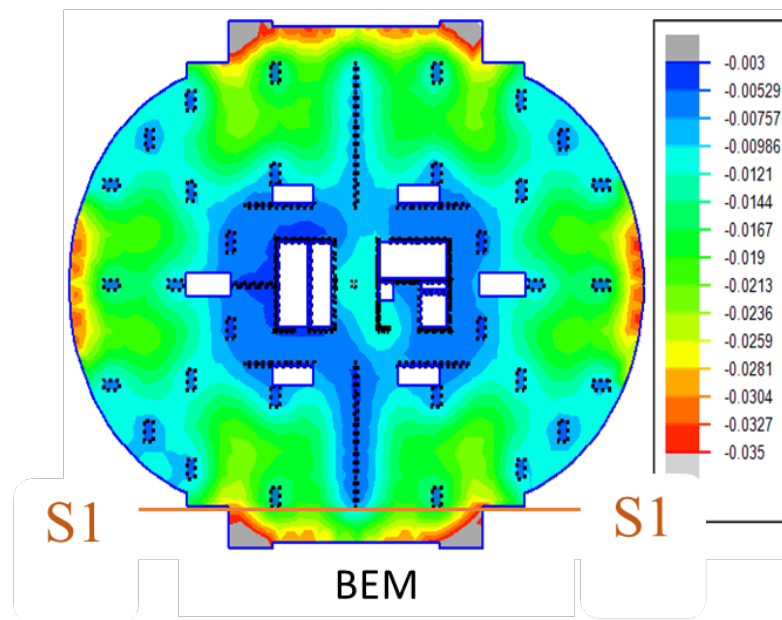
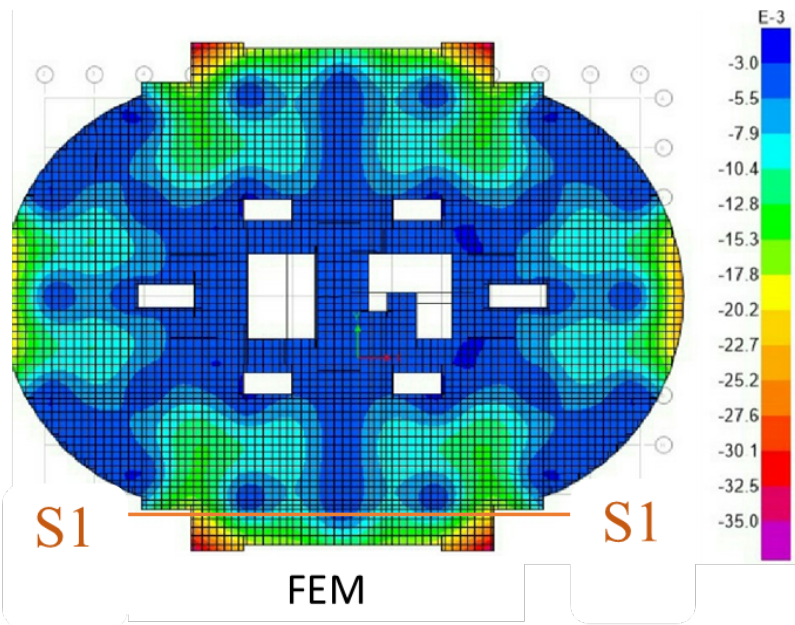


Revit analytical model



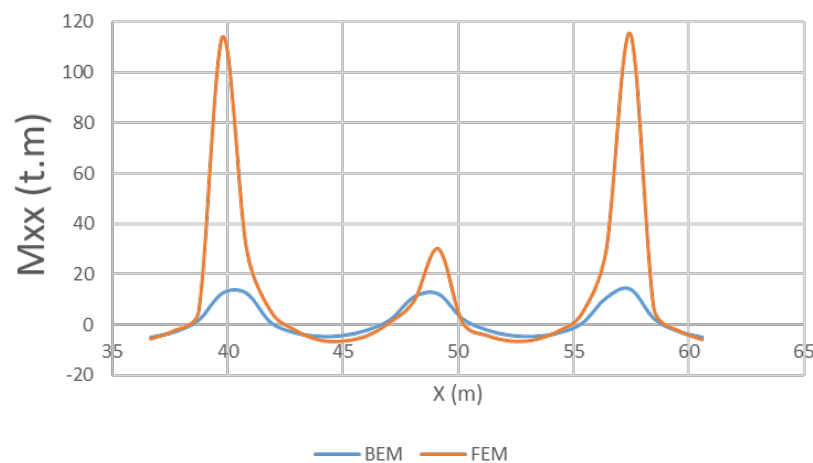
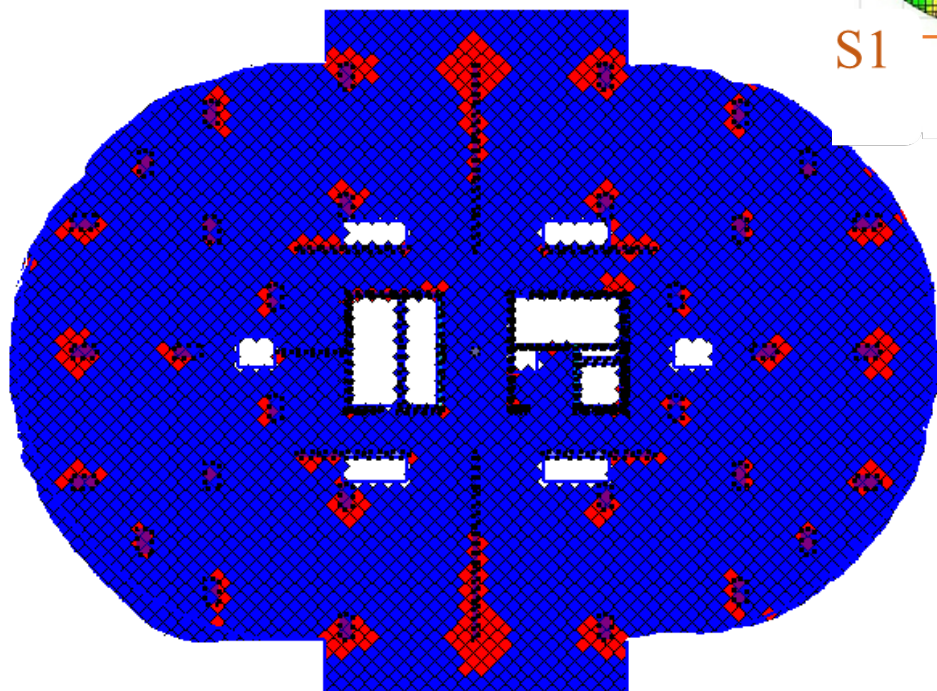
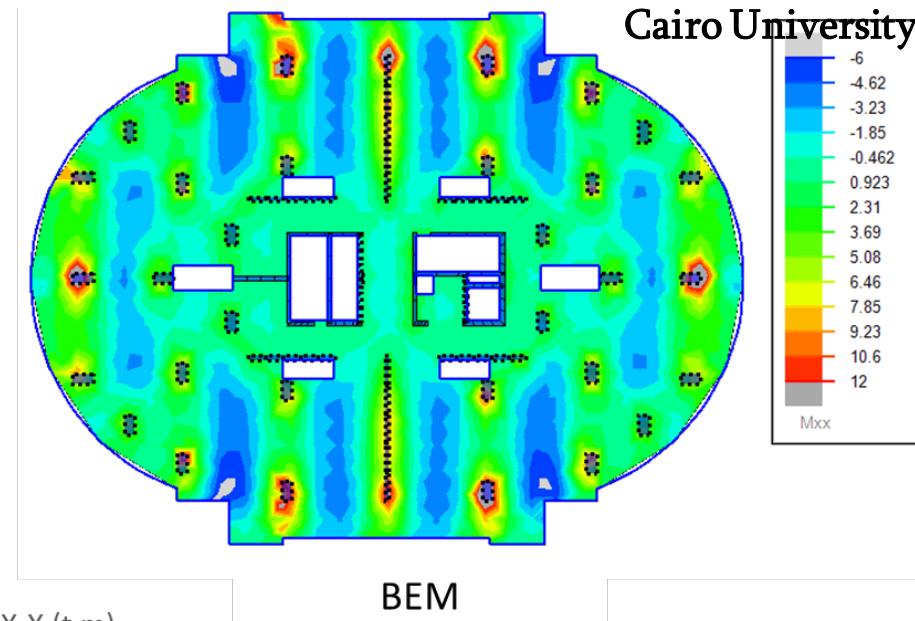
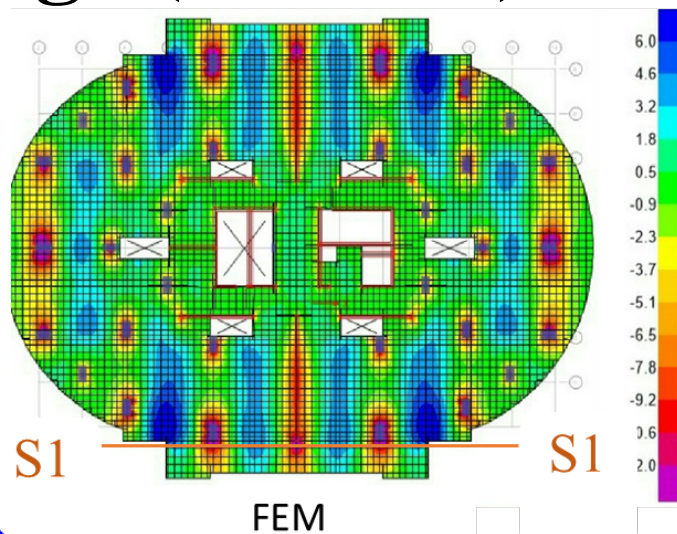
Revit geometrical model with vertical loads

5. Fixed base Package (FBPAK)



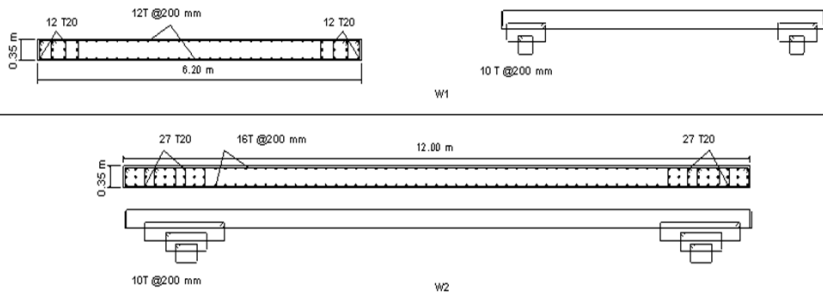
PLPAK 3D model

5. Fixed base Package (FBP)

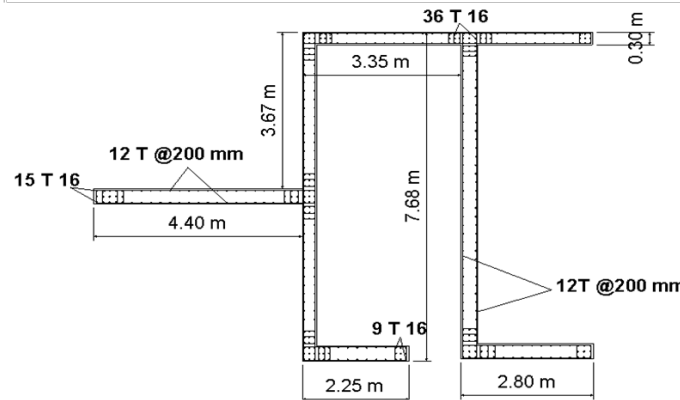


5. Fixed base Package (FBP)

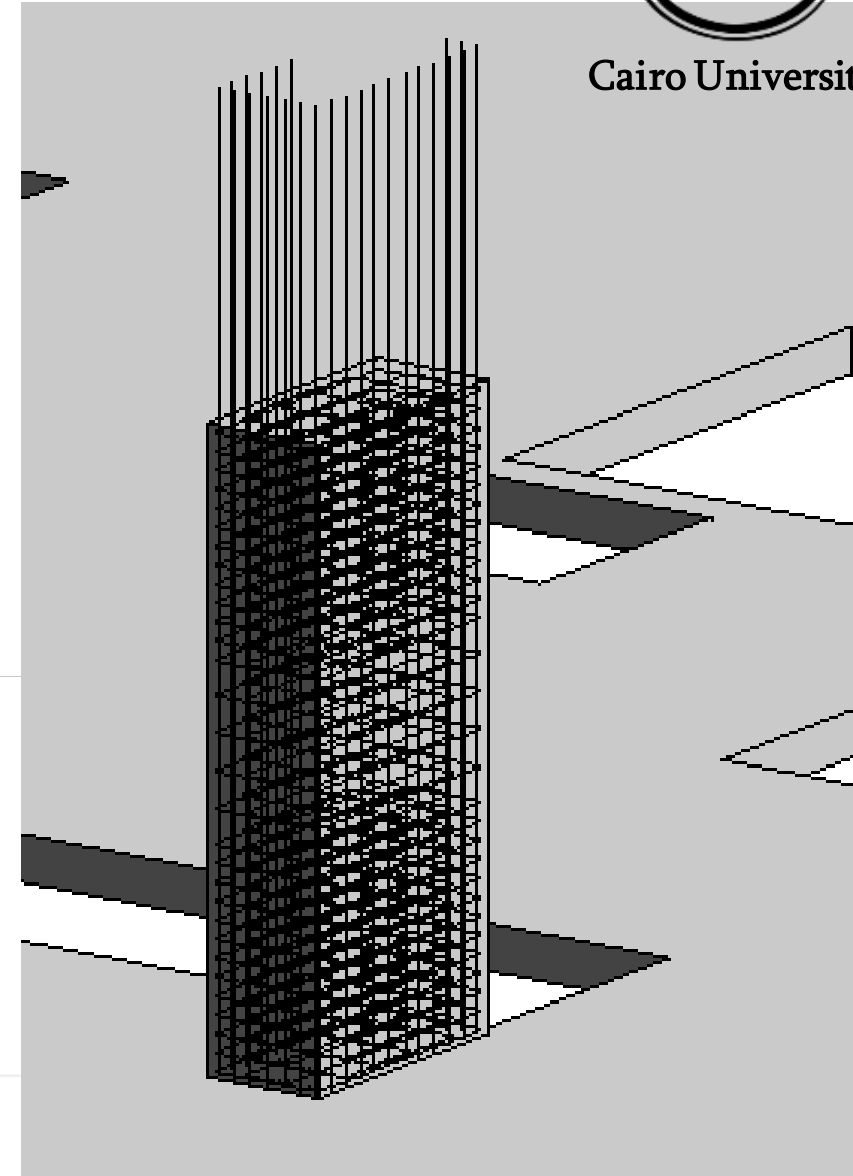
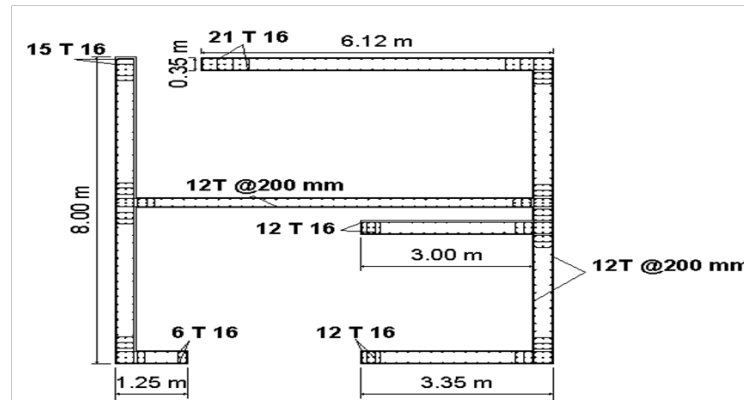
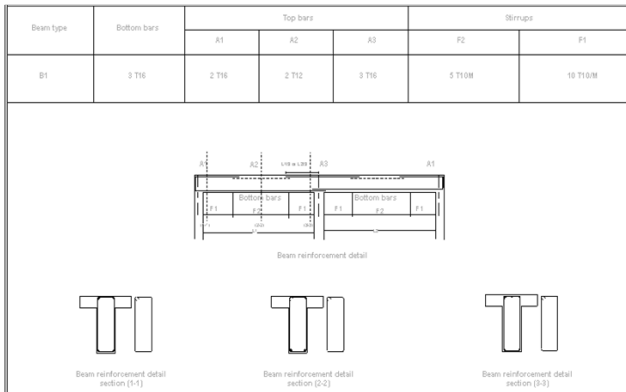
Design of Shear walls:



Design of Cores:

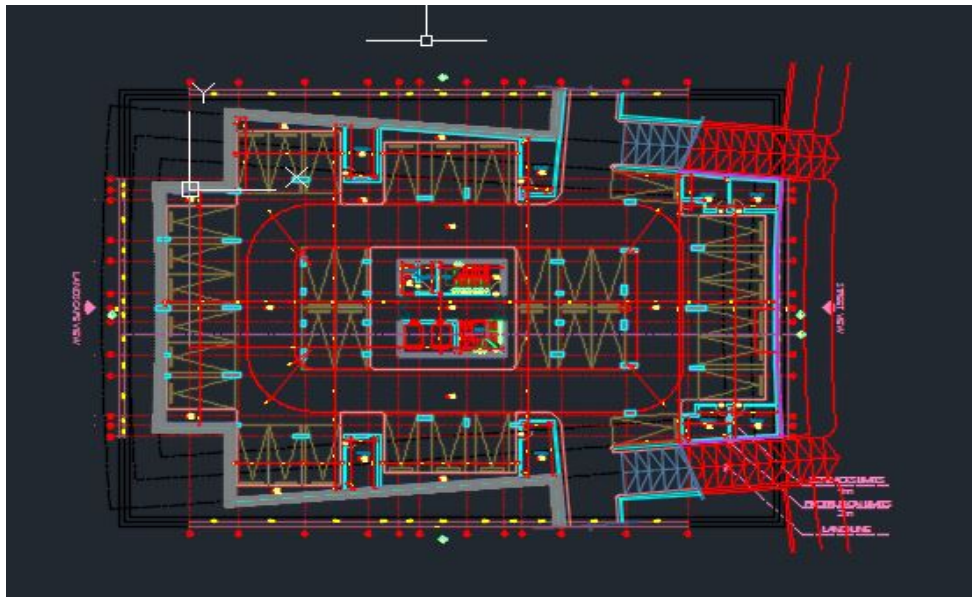


Design of beams:

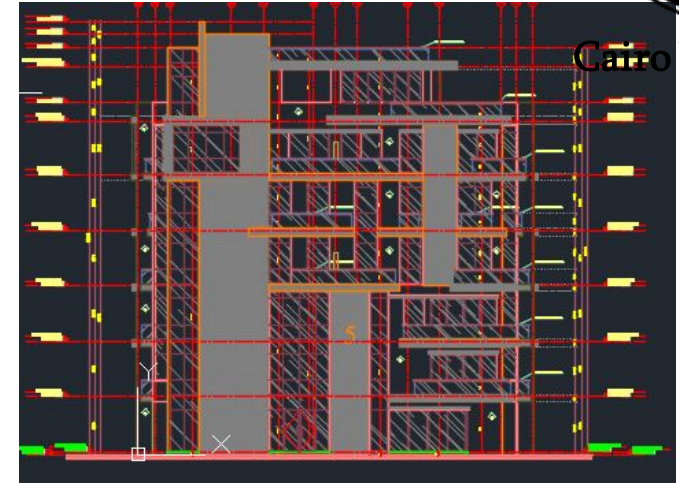


5. Fixed base Package (FBP)

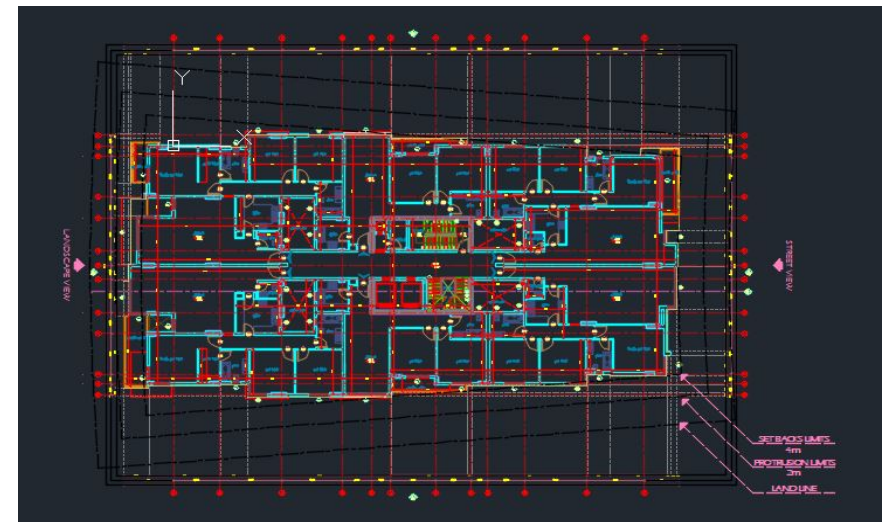
The project is a residential building, consisting of 8 different floors.
The average area of slabs = 770m² and the height = 2.97m, for basement slab = 1150m² and the height equals 3.06m



Architectural plan for the basement

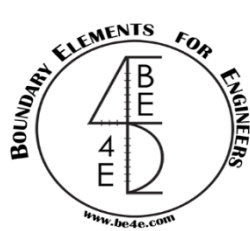


Elevation of the building



Architectural plan for the first floor.

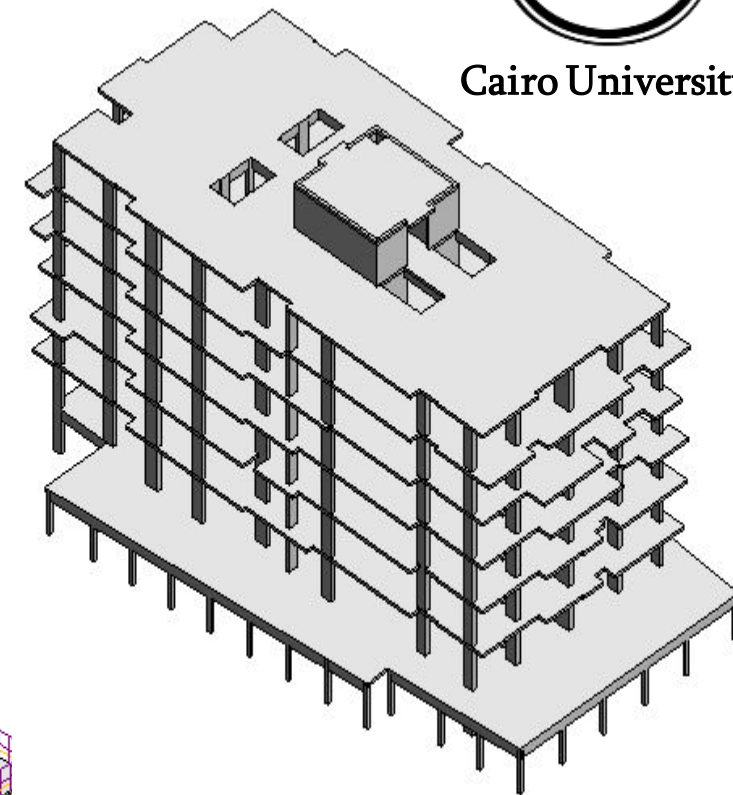
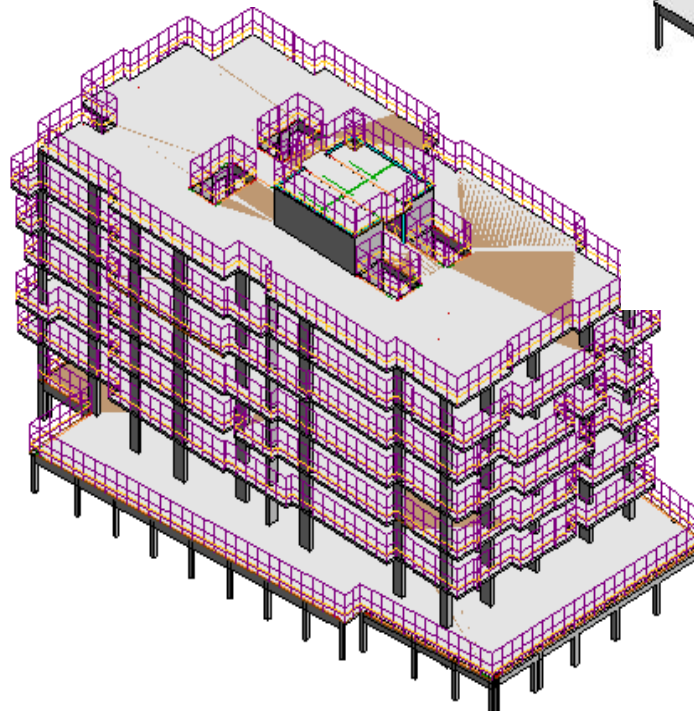
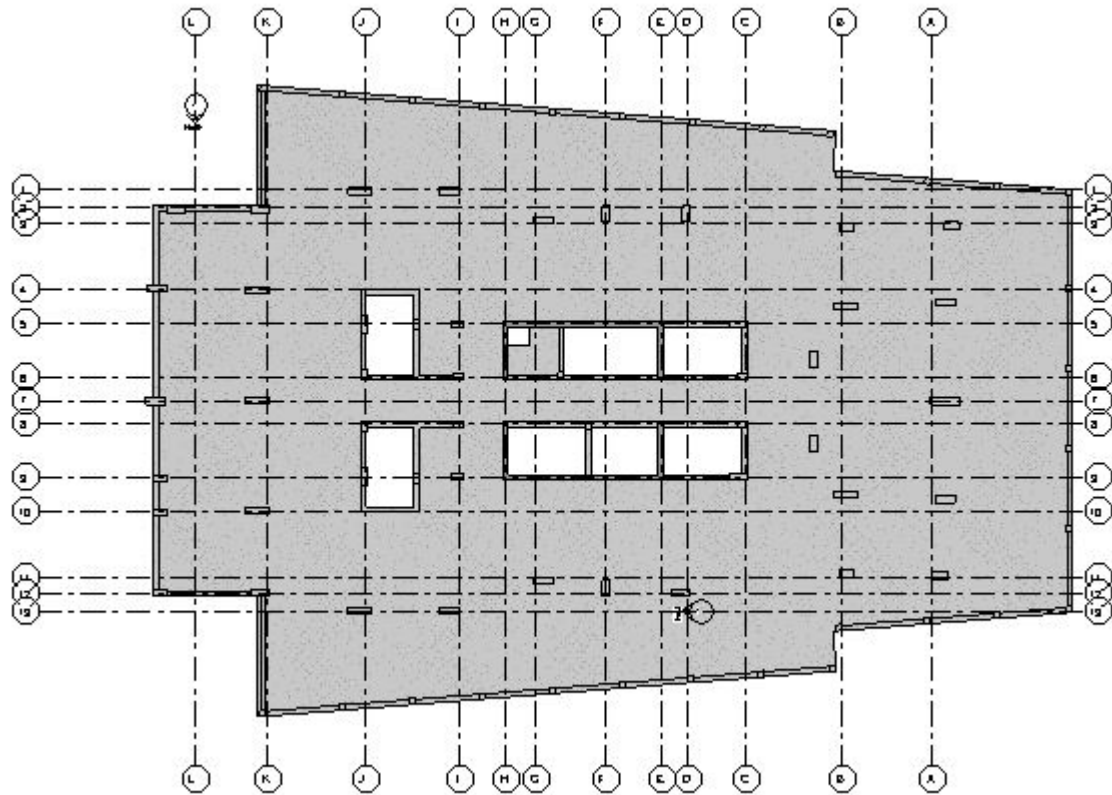
CUBE-BE



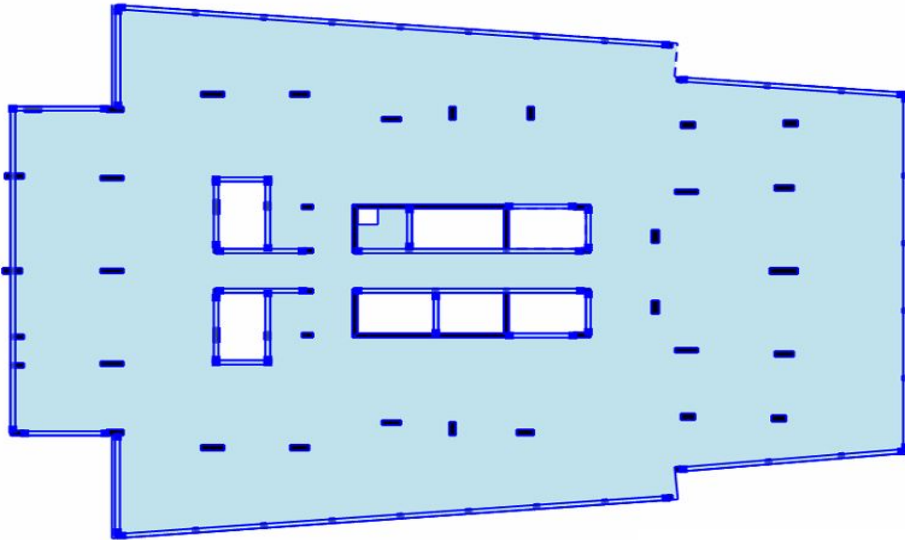
Cairo University

5. Fixed base Package (FBP)

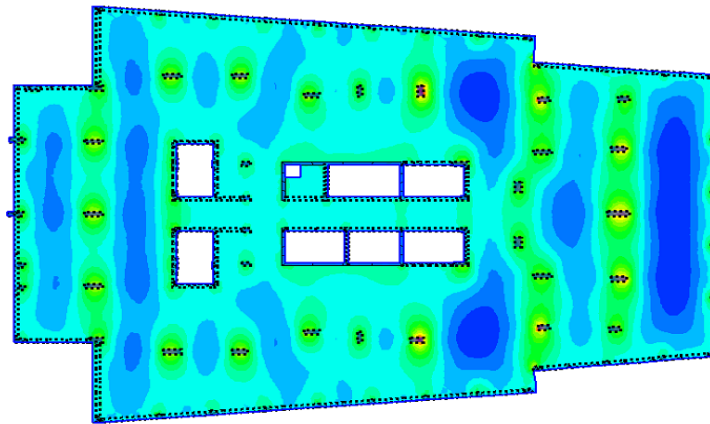
www.be4e.com



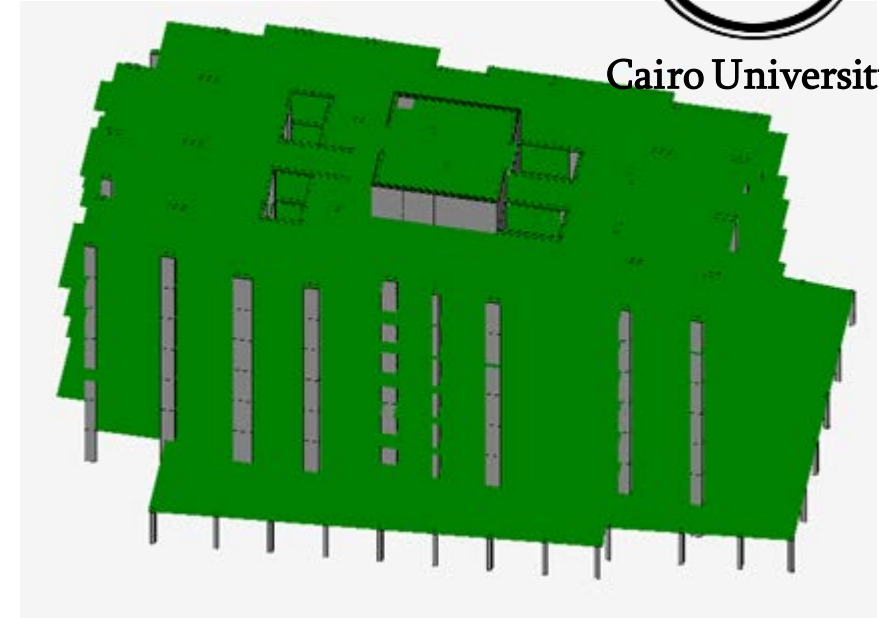
5. Fixed base Package (FBP)



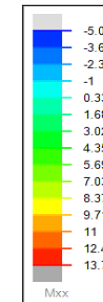
Basement Floor in PLGen



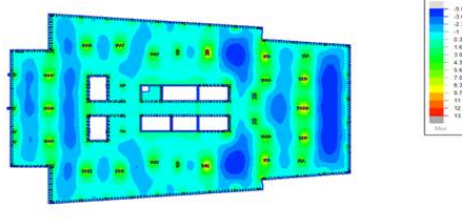
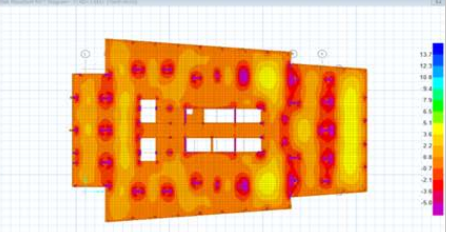
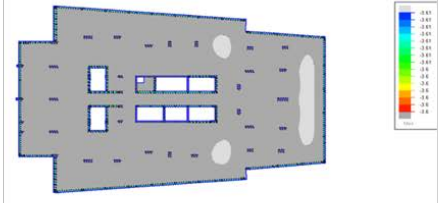
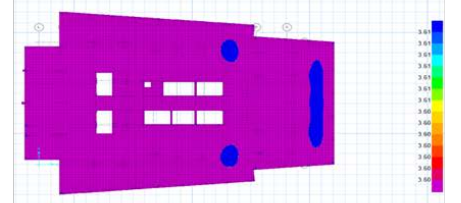
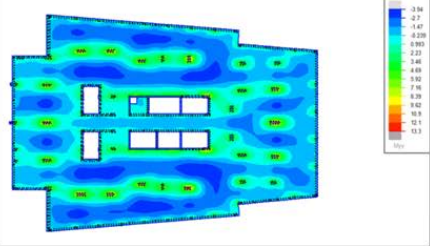
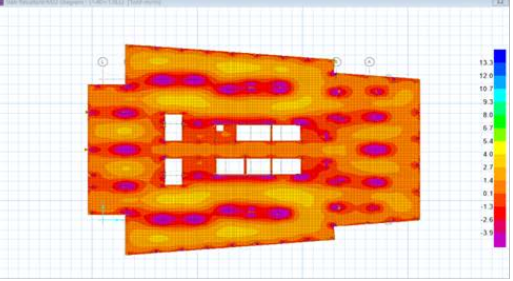
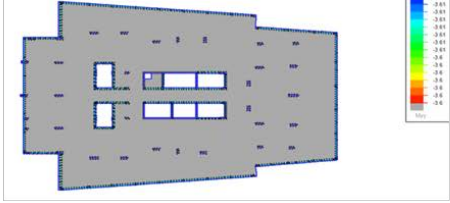
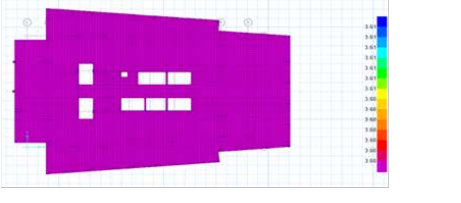
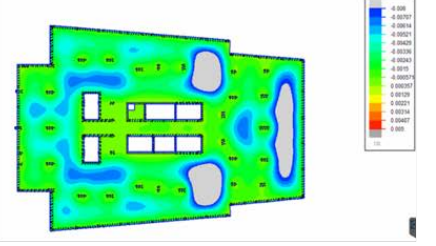
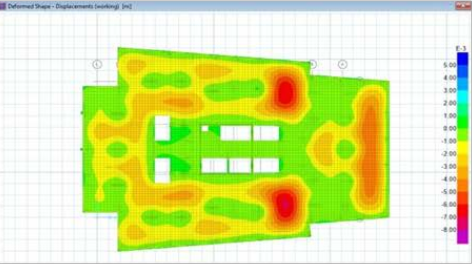
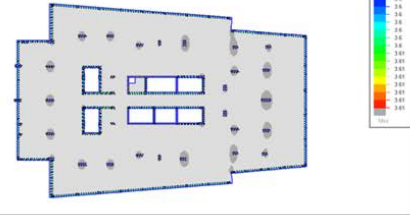
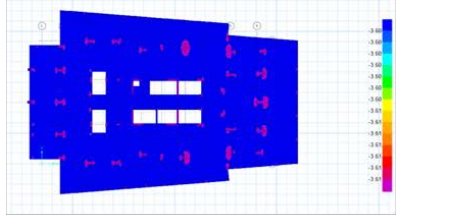
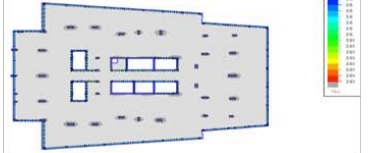
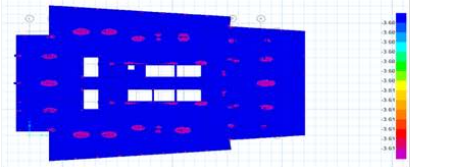
Bending moment M_{xx} of Basement Floor in PLPOST



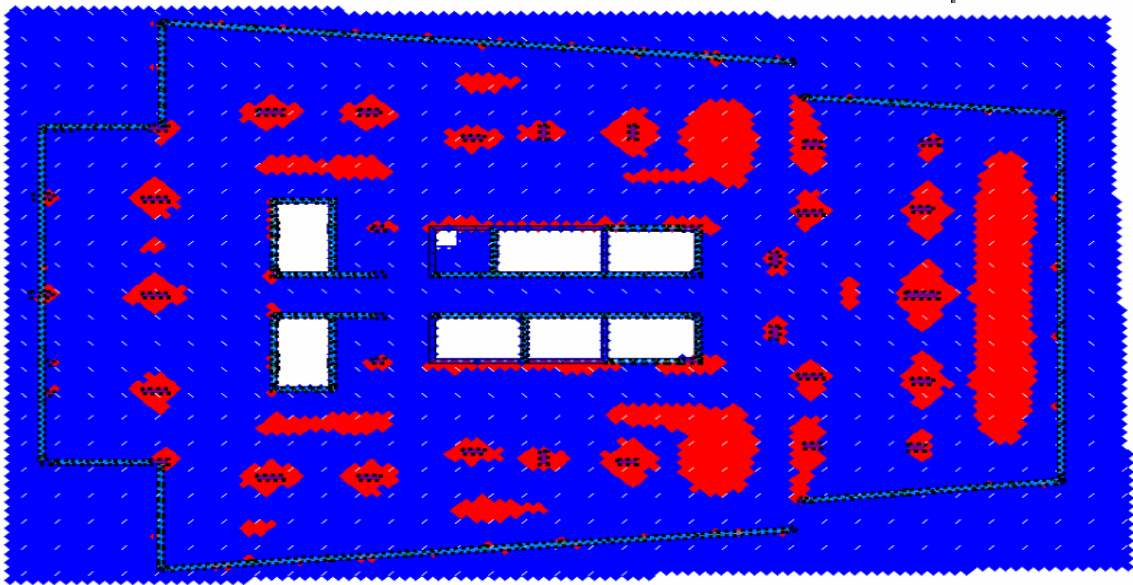
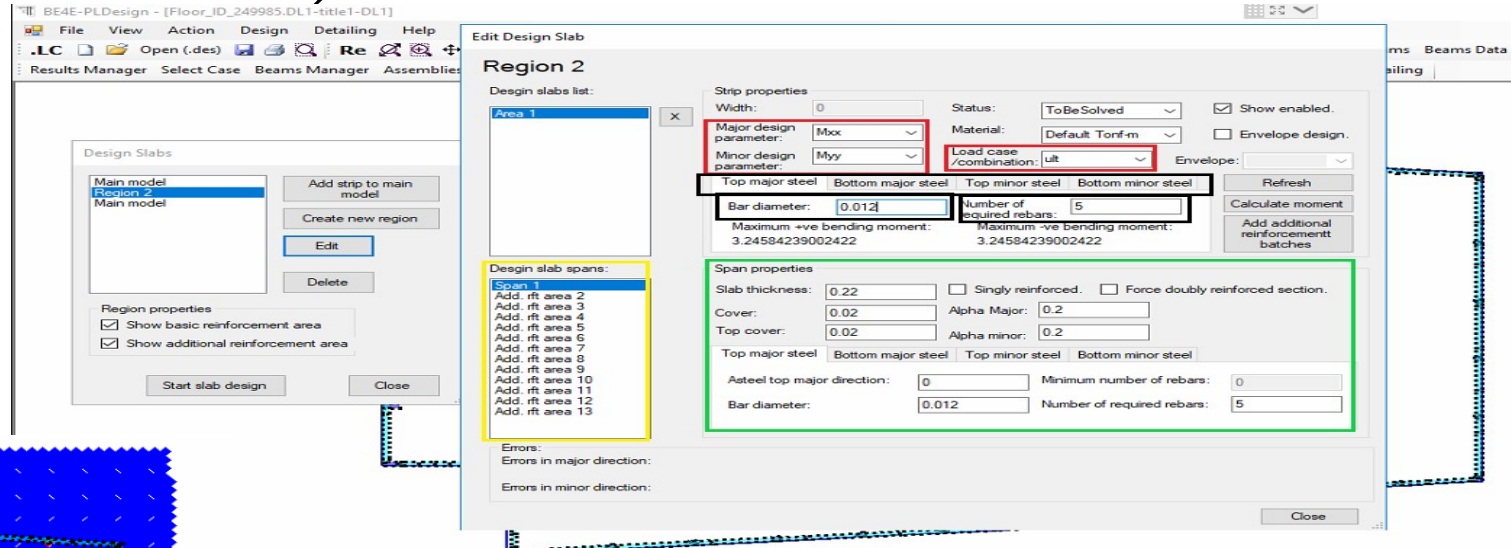
3D PLPAK model



5. Fixed base Package (FB) www.be4e.com

	PLPOST (BE)	SAFE (FE)		PLPOST (BE)	SAFE (FE)
Mxx			Mxx (B)		
Myy			Myy (B)		
Def.			Mxx (T)		
			Myy (T)		

5. Fixed base Package (FBP) www.be4e.com

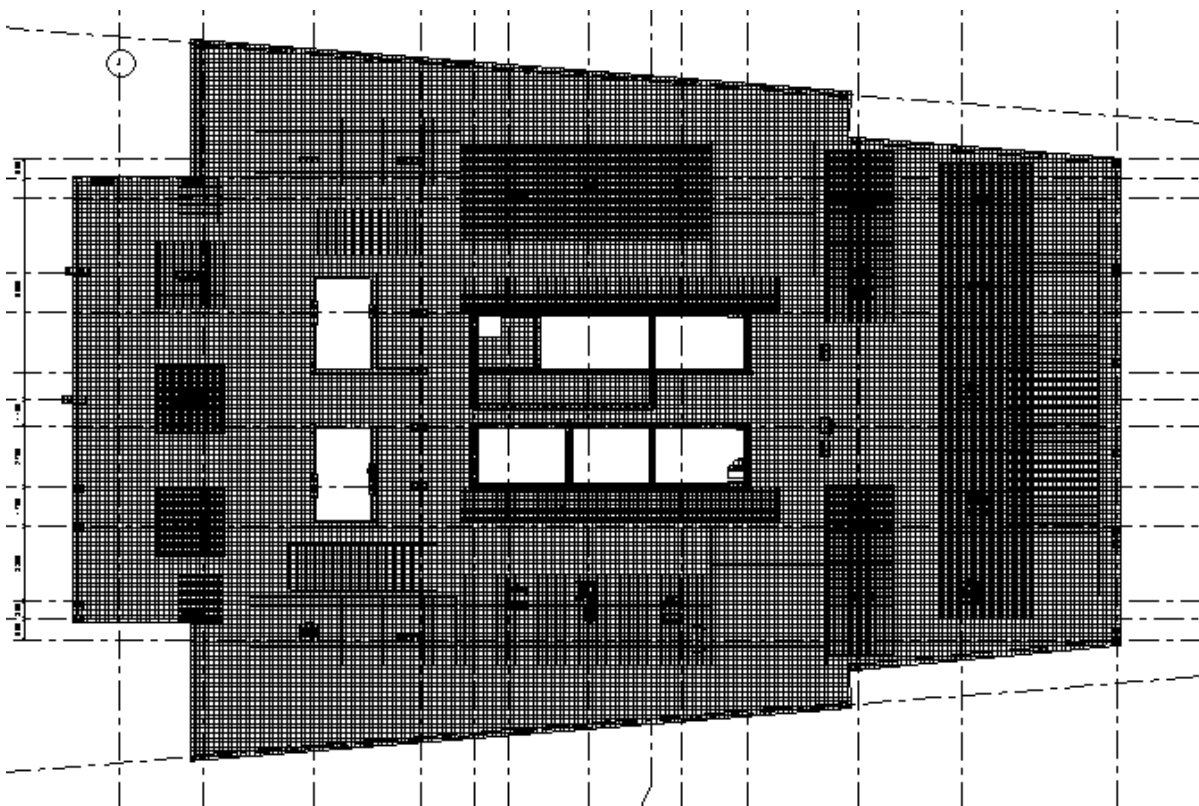


Basement slab design in PLDesign

5. Fixed base Package (FBP)

www.be4e.com

Cairo University



Define reinforcement details

Design Beam10 - at sup1 reinforcement details

Straining actions
Design cell: 175 Design beam element: 64
Flexure design load case/combination: ult Design moment: -2.24152088165
Shear design load case/combination: ult Design shear: -9.52139377593
Torsion design load case/combination: ult Design torsion: -0.11654066294

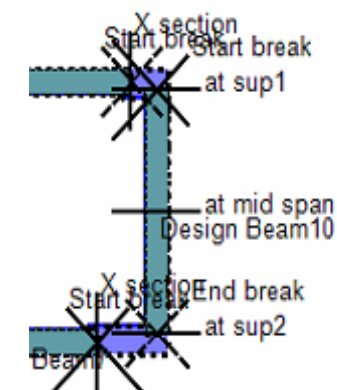
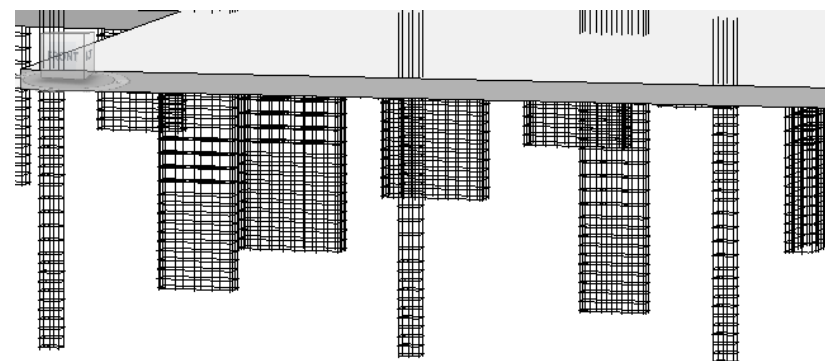
NoOfBars	BarDiameter	depth	IsBottomLayer
3	0.012	0	<input type="checkbox"/>
2	0.012	0	<input checked="" type="checkbox"/>

Required Asteel top: 0 Actual Asteel top: 0.000339292006
Required Asteel bottom: 0 Actual Asteel bottom: 0.000226194671
Bending bottom: Safe
Bending top: Safe
Stirrups
Width: 0.2 NoOfLegs: 2 BarDiameter: 0.008 BarSpacing: 0.01
Required Asteel: 0 Actual Asteel: 0.010053096491
Stirrups (shear-torsion): Safe

Cover top: 0.025
Cover left: 0.025
Cover right: 0.025
Cover bottom: 0.025
Refresh

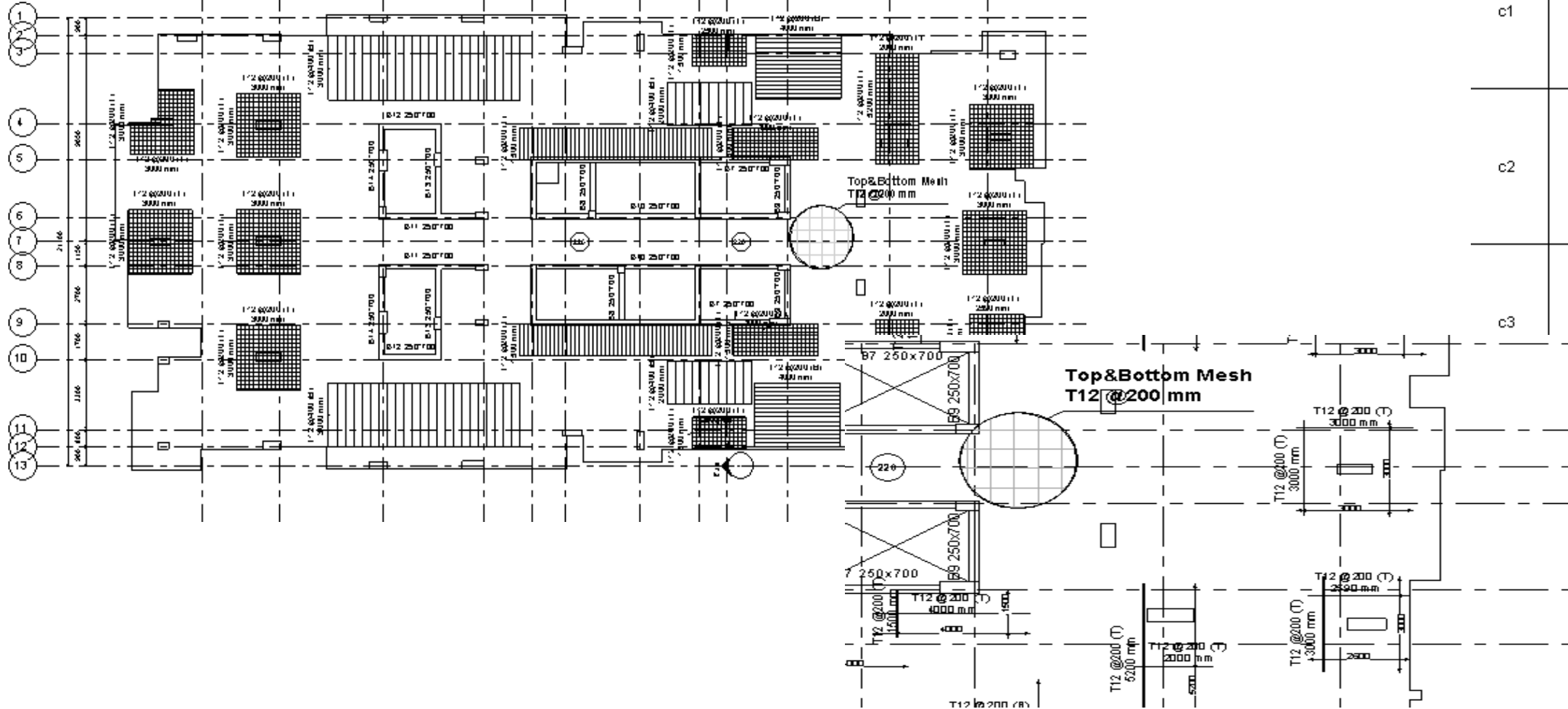
BarDiameter	Xbar	Ybar
0.012	0.035	0.35
0.012	0.215	0.35

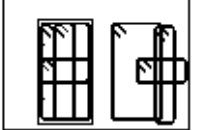
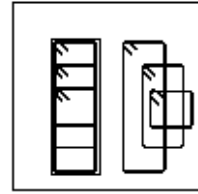
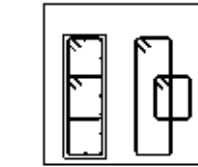
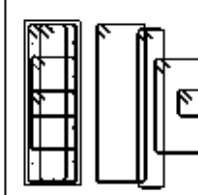
Required Asteel: 0 Actual Asteel: 0.000226194671
Longitudinal bars torsion: Safe
Add longitudinal bar
Close



5. Fixed base Package (FBP)

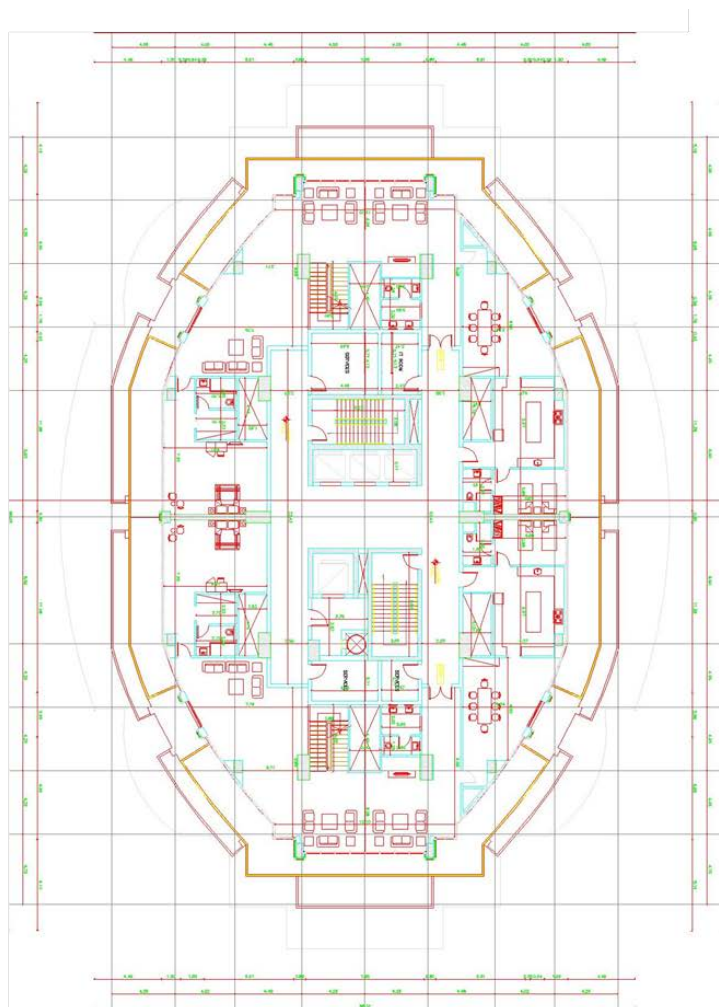
www.be4e.com



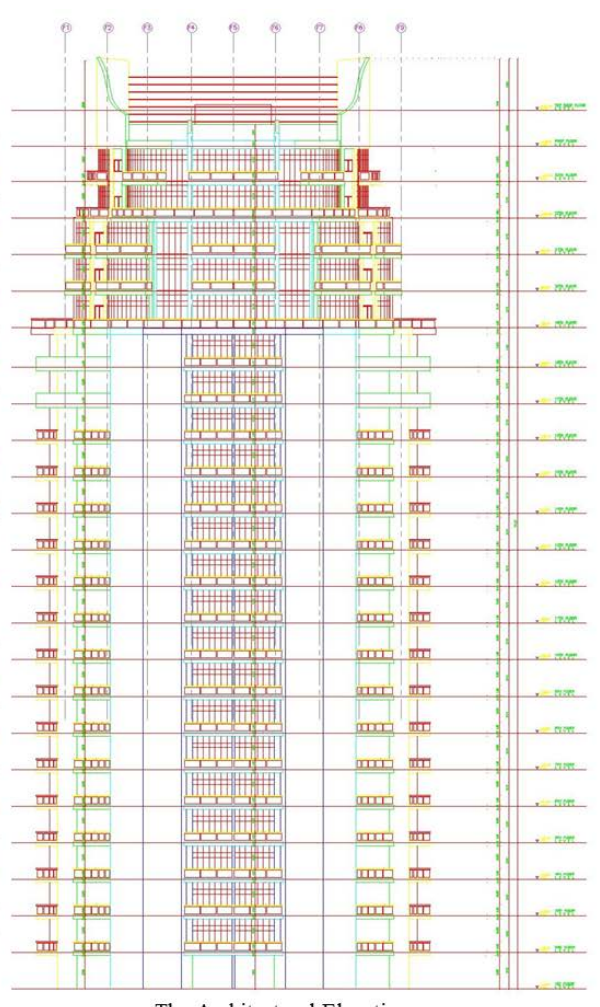
column	section (scale 1:25)
c1	
c2	
c3	
	

Cairo University

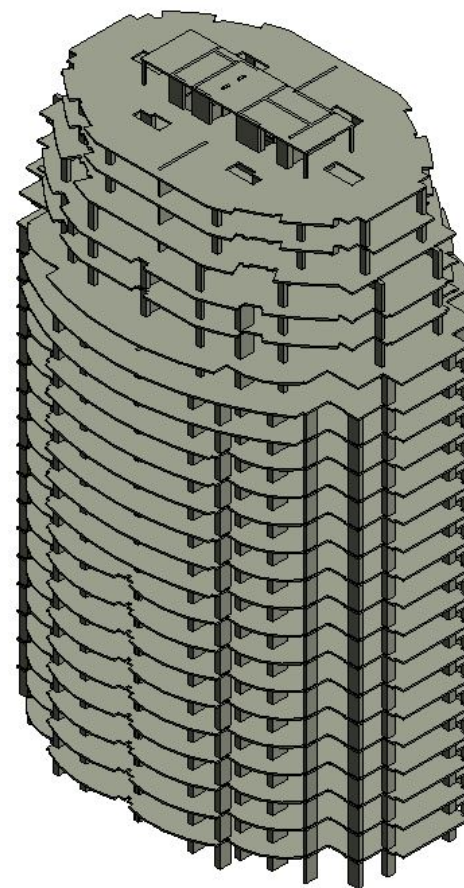
5. Fixed base Package (FBP)



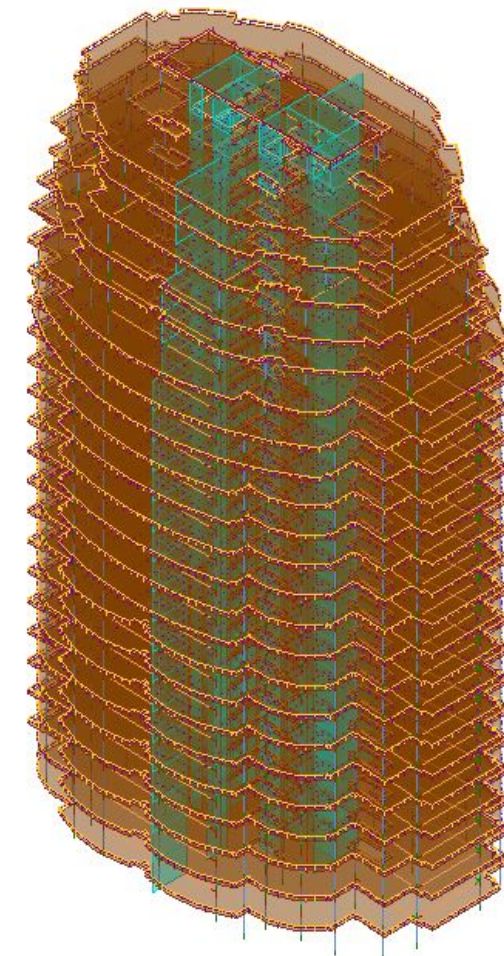
The Architectural Plan of the 22nd Floor.



The Architectural Elevation.



Revit 3D geometric Model

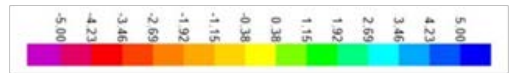
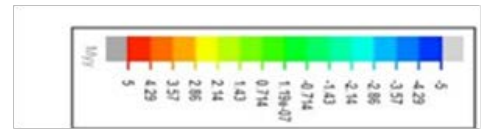
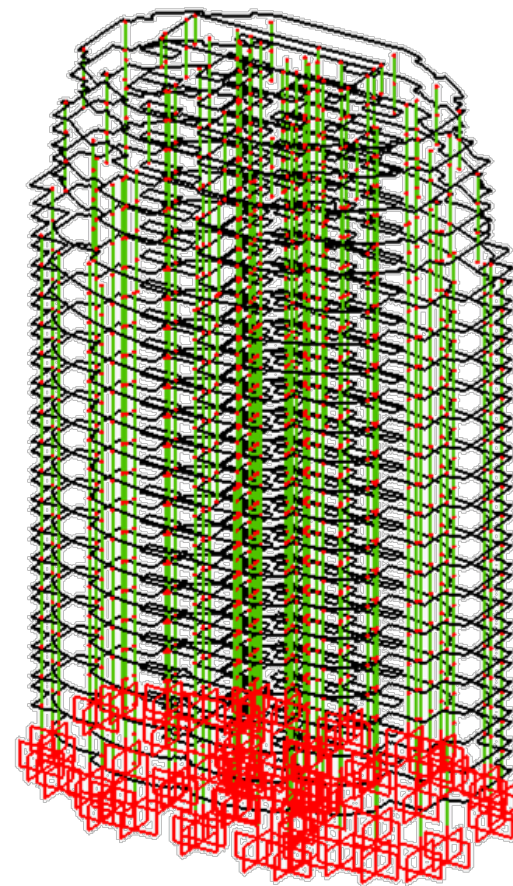
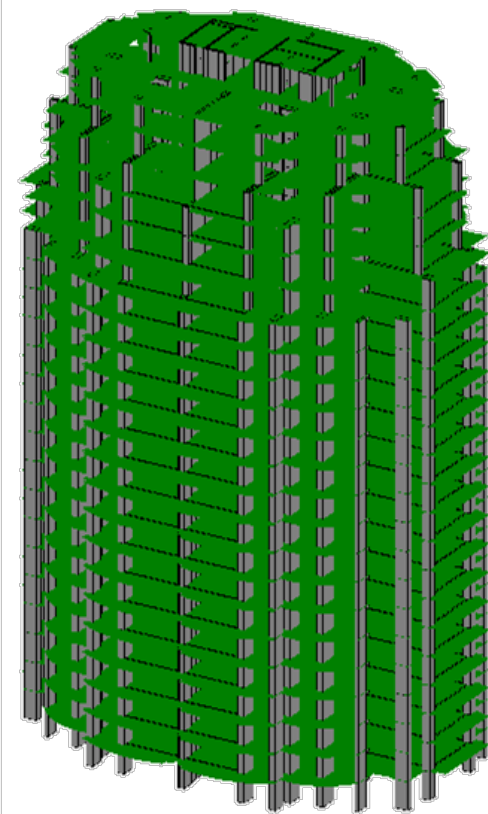
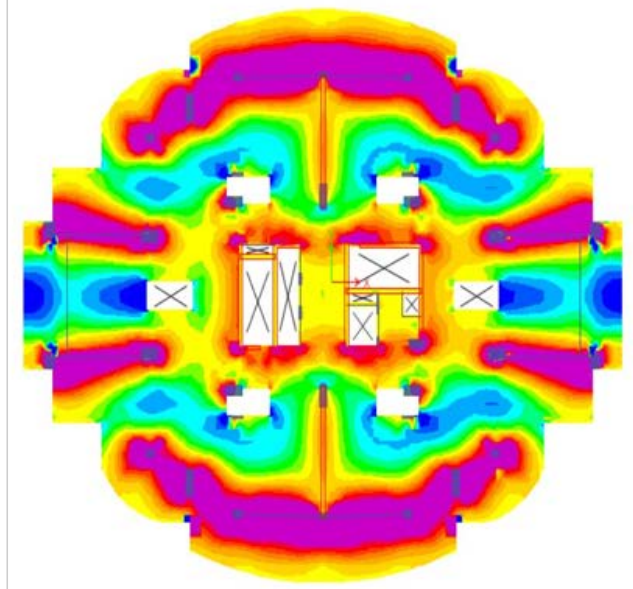
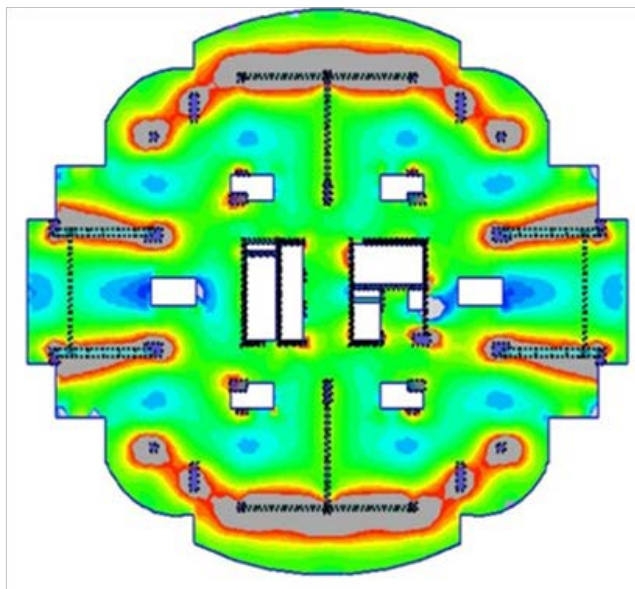


Revit 3D analytical Model

5. Fixed base Package (FBP)

www.be4e.com

Cairo University



The M_{yy} 's contour for the 19th floor on BE tool "PLPAK" in t.m.

The M_{yy} 's contour for the 19th floor on FE tool "SAFE" in t.m.

PLPAK 3D geometric Model

PLPLAK 3D analytical Model

5. Fixed base Package (FBP)

Edit Design Slab

Region 2

Design slabs list:

- Area 1

Strip properties

Width: 0 Status: Solved Show enabled.

Major design parameter: Mxx Material: Default Tonf-m Envelope design.

Minor design parameter: Myy Load case /combination: ULTIMATE Envelope:

Top major steel Bottom major steel Top minor steel Bottom minor steel Refresh

Bar diameter: 0.012 Number of required rebars: 5 Calculate moment

Maximum +ve bending moment: 5.01025117921679 Maximum -ve bending moment: 5.01025117921679 Add additional reinforcement batches

Design slab spans:

- Span 1
- Add. rft area 2
- Add. rft area 3
- Add. rft area 4
- Add. rft area 5
- Add. rft area 6

Span properties

Slab thickness: 0.25 Singly reinforced. Force doubly reinforced section.

Cover: 0.025 Alpha Major: 0.2

Top cover: 0.025 Alpha minor: 0.2

Top major steel Bottom major steel Top minor steel Bottom minor steel

Asteel top major direction: 0.00056548667 Minimum number of rebars: 5

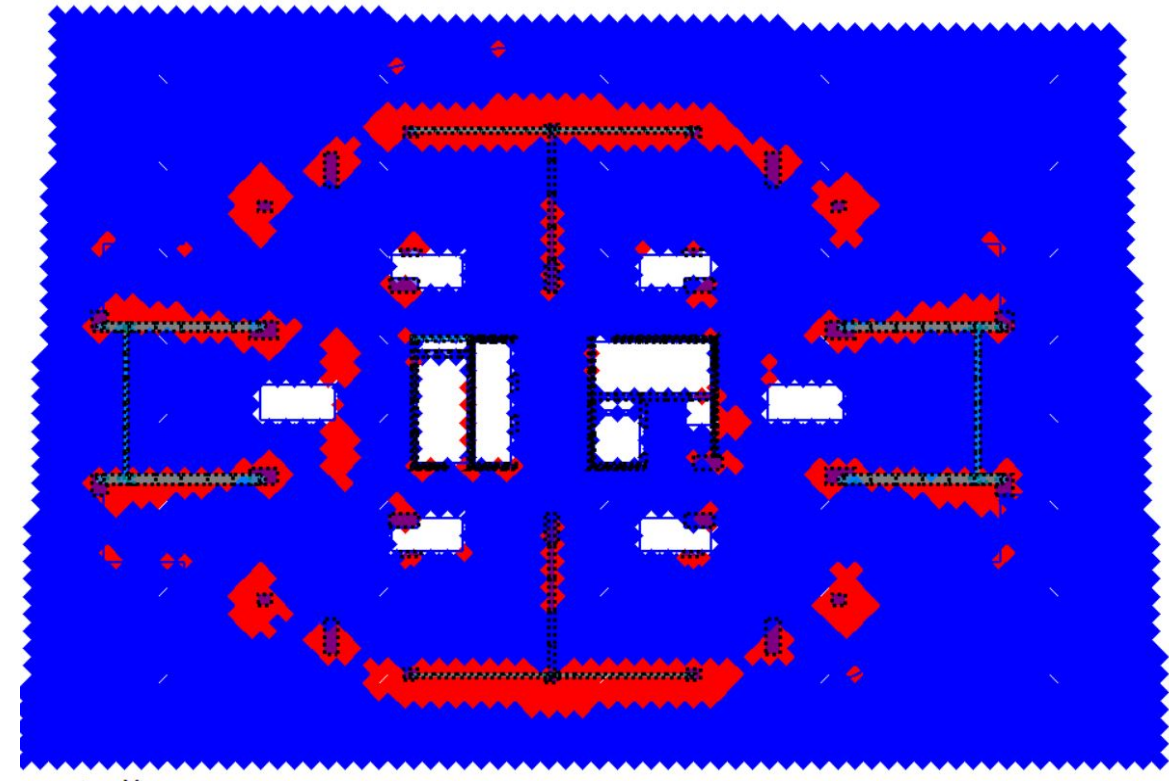
Bar diameter: 0.012 Number of required rebars: 5

Errors:

Errors in major direction:

Errors in minor direction:

Close

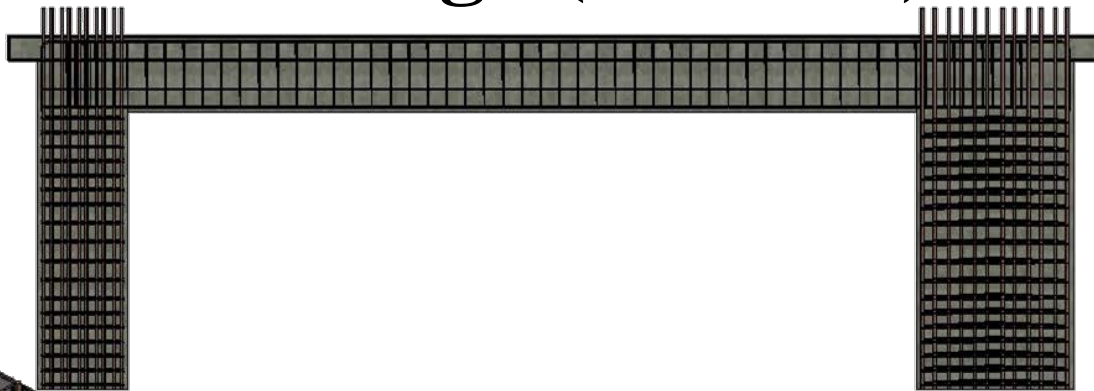


The moment contour on PLGen where additional RFT is needed

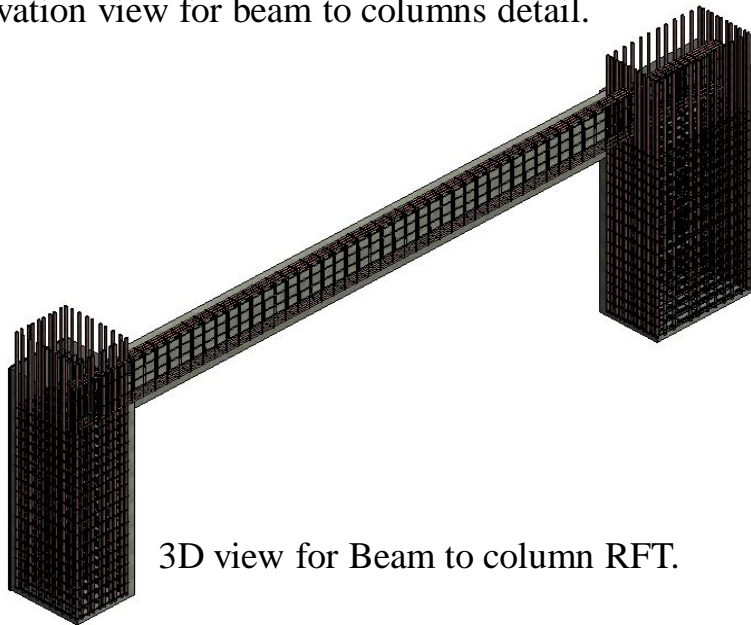
The slab design tool where RFT is generated.

5. Fixed base Package (FBP)

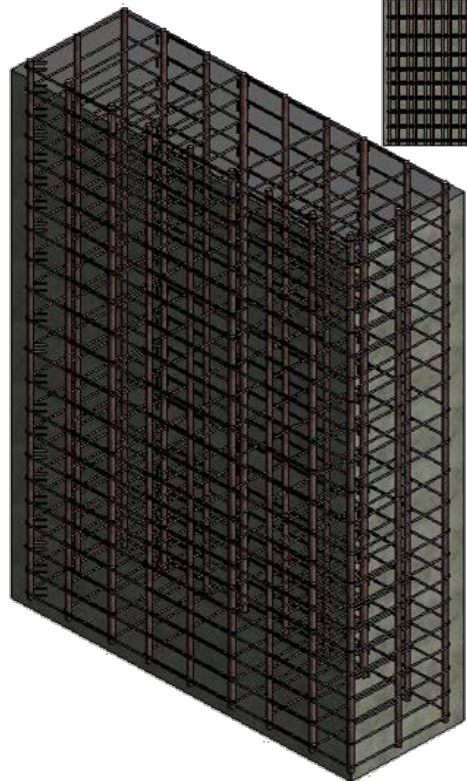
www.be4e.com



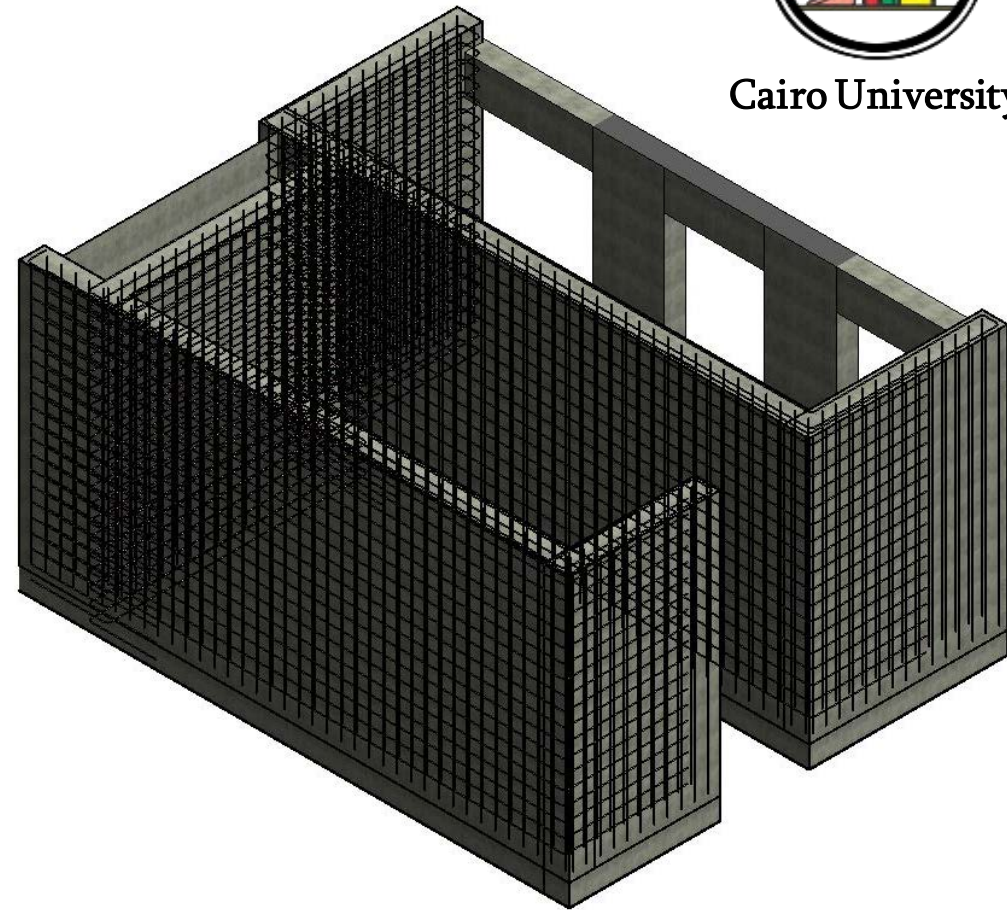
Elevation view for beam to columns detail.



3D view for Beam to column RFT.



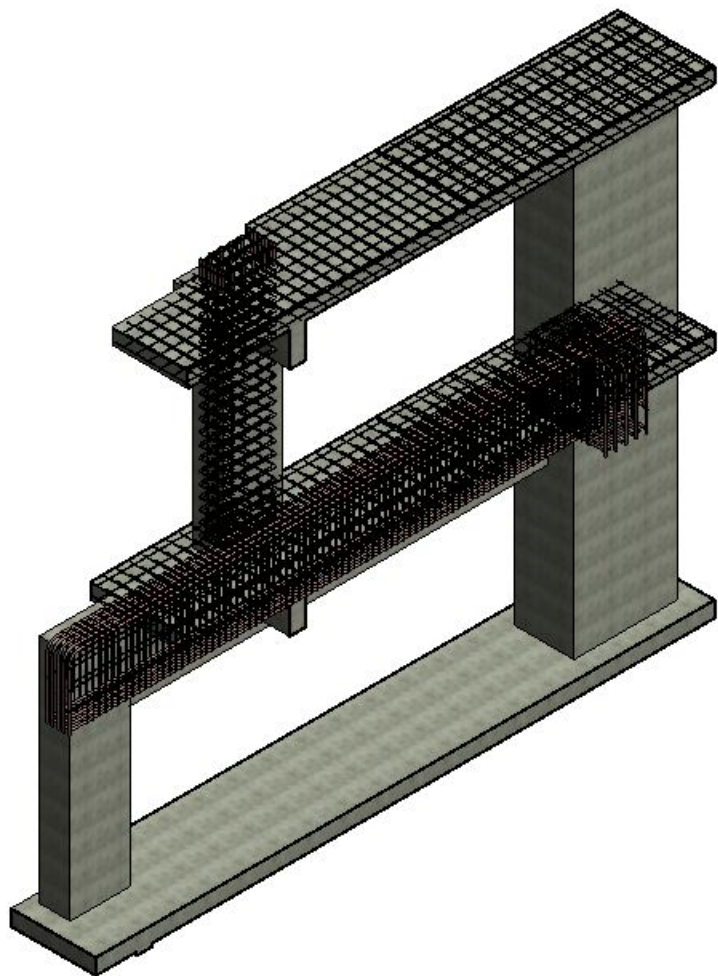
3D view for column RFT detail.



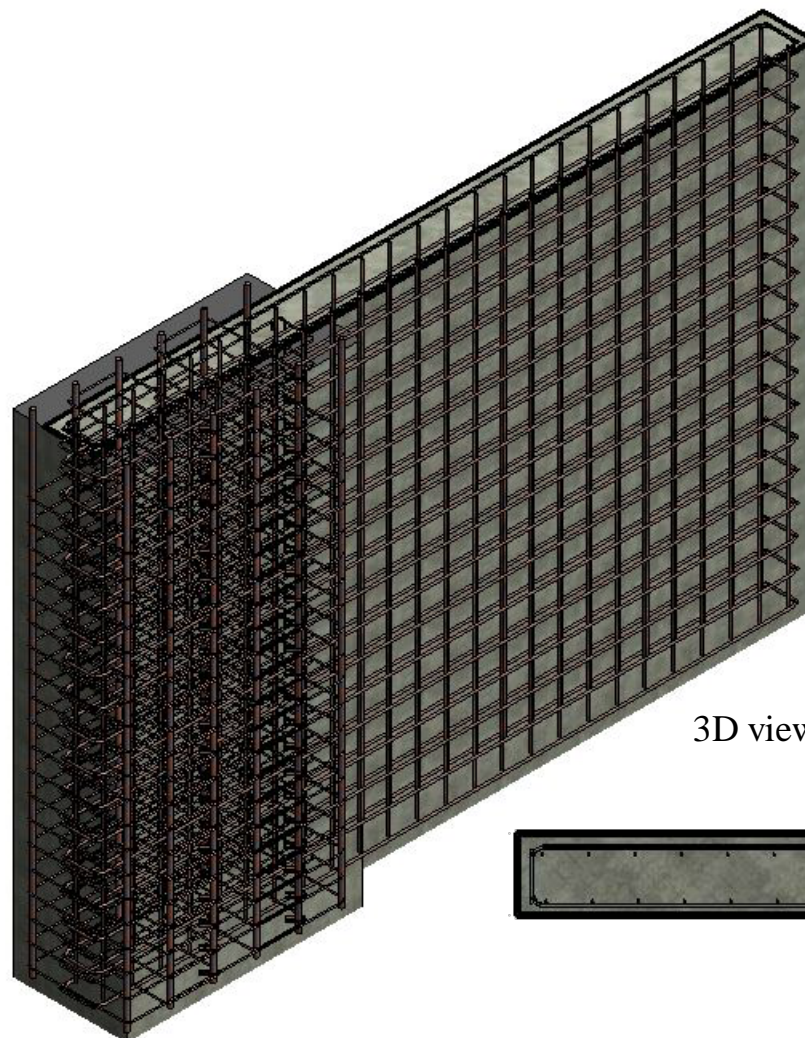
3D view for core RFT detail.

5. Fixed base Package (FBP)

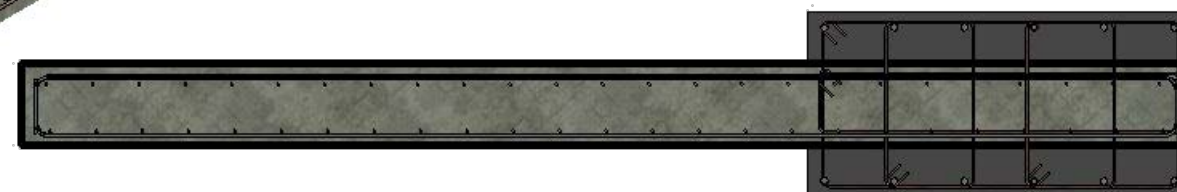
www.be4e.com



3D Detailing between beams and above column.



3D view for shear wall detail.



x-section for shear wall Detail.

CUFE-BE



www.be4e.com



Cairo University

Table of contents

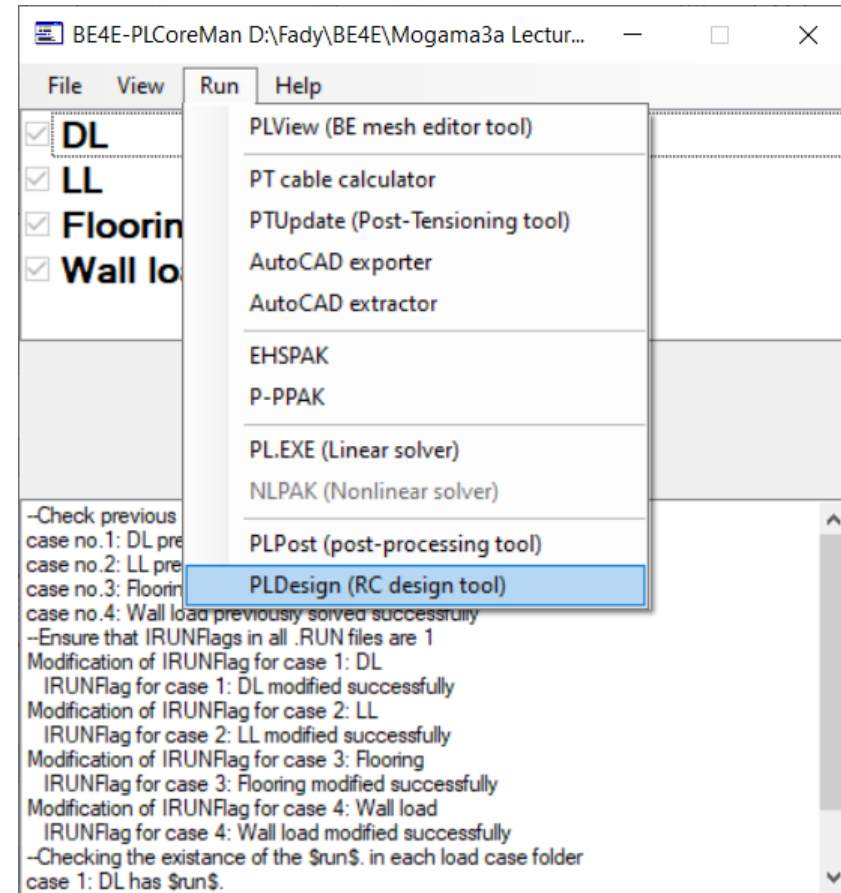
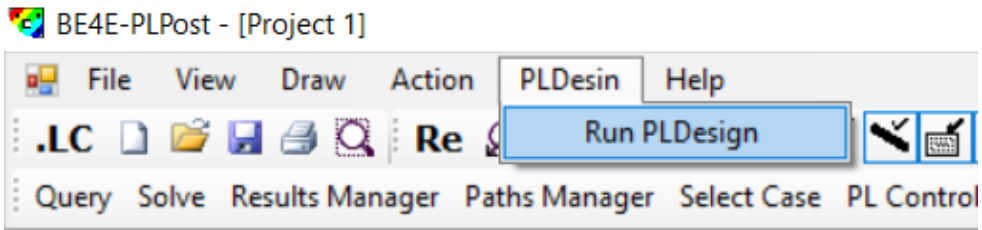
- 1. Introduction**
- 2. Basic package (PLPAK Basic)**
- 3. One floor package (BIM-PLPAK)**
- 4. Foundation Package (FoundPAK)**
- 5. Fixed base Package (FBPAK)**
- 6. Post-tension Package (PTPAK)**
7. Dynamic package (DynPAK)
8. Overall building package (OBPAK)
9. 4D and 5D analysis
10. Conclusions

What is the PTPAK?

- PTPAK (Post tension package) is a structural design tool package for post-tensioned plate bending structures based on the boundary element method for shear deformable plate bending theory, using different codes like (ACI, EC, and ECP).
- The PTPAK is added to the PLPAK-Basic package to design reinforced concrete building slabs and foundations.
- The PTPAK is not only consider about design, but also about detailing and calculation sheet forming.
- In PTPAK the user can change the cable profile (13 templates) to serve the different structure conditions.
- In PTPAK the user can draw reinforcement on slab and determine the reinforcement at beams (without post tension).
- In the PTPAK the calculation is not only for load combination, but also for envelopes.

What is the PTPAK?

- The user can go to PTPAK by two ways either by using PLPost or PLCoreMan as follows:



6. PTPAK Package

6.1 Files need to be exported before using PTPAK

6.2 How can users generate PTPAK Model

6.2.1 Draw cables

6.2.2 Cables data

6.2.3 Cable templates

6.2.4 Multiple cable selection

6.3 How to solve the PTPAK model

6.4 Load combinations & load envelopes

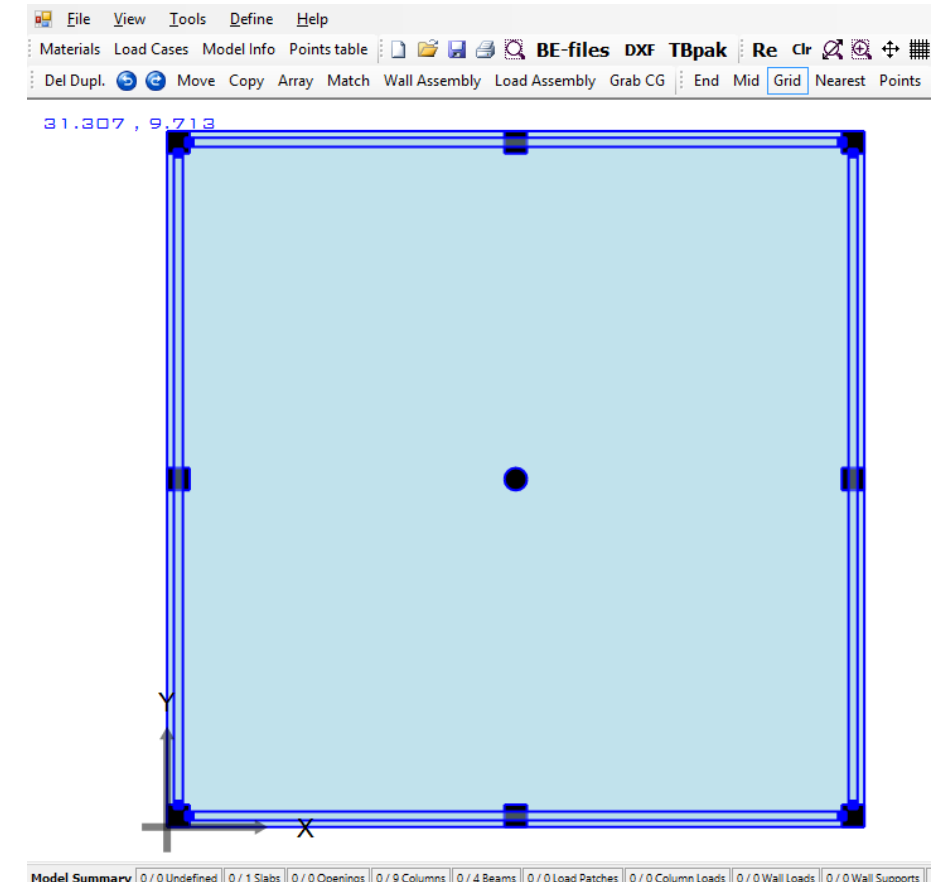
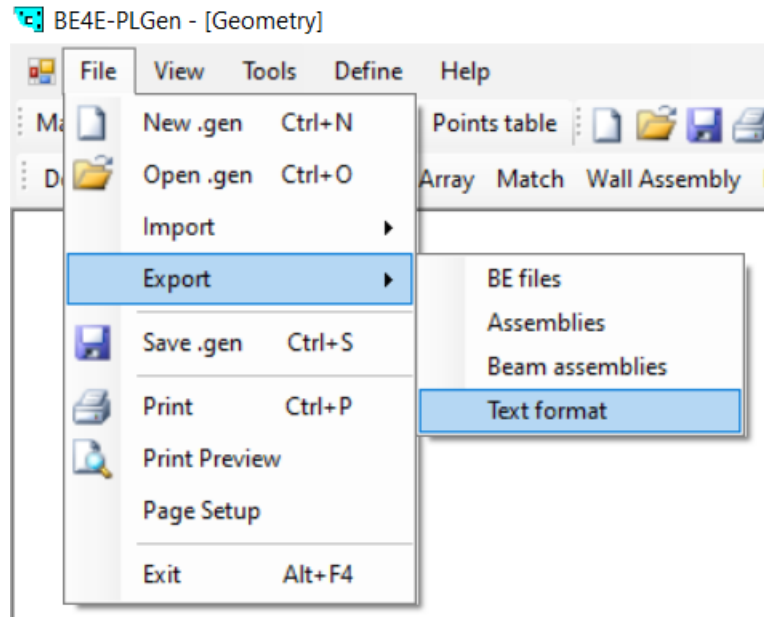
6.5 Check the Cable Eccentricity

6.6 Check the Cable stresses

6.7 Optimizer

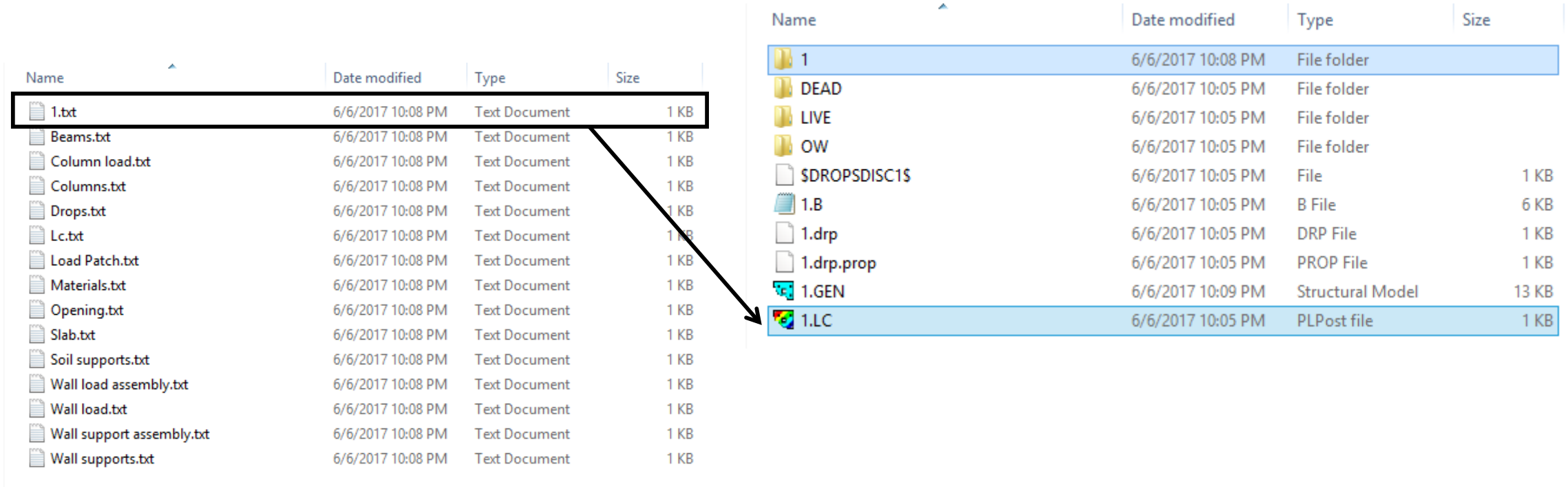
6.1. File needed to be exported before using PTPAK

After drawing the problem (without post tension) in PLGen as shown below BE-Files should be exported and the text files also for the problem.



6.1. File needed to be exported before using PTPAK

N.B. the text format should named similarly as .LC file in a created folder with also the same name of .LC file.



Name	Date modified	Type	Size
1.txt	6/6/2017 10:08 PM	Text Document	1 KB
Beams.txt	6/6/2017 10:08 PM	Text Document	1 KB
Column load.txt	6/6/2017 10:08 PM	Text Document	1 KB
Columns.txt	6/6/2017 10:08 PM	Text Document	1 KB
Drops.txt	6/6/2017 10:08 PM	Text Document	1 KB
Lc.txt	6/6/2017 10:08 PM	Text Document	1 KB
Load Patch.txt	6/6/2017 10:08 PM	Text Document	1 KB
Materials.txt	6/6/2017 10:08 PM	Text Document	1 KB
Opening.txt	6/6/2017 10:08 PM	Text Document	1 KB
Slab.txt	6/6/2017 10:08 PM	Text Document	1 KB
Soil supports.txt	6/6/2017 10:08 PM	Text Document	1 KB
Wall load assembly.txt	6/6/2017 10:08 PM	Text Document	1 KB
Wall load.txt	6/6/2017 10:08 PM	Text Document	1 KB
Wall support assembly.txt	6/6/2017 10:08 PM	Text Document	1 KB
Wall supports.txt	6/6/2017 10:08 PM	Text Document	1 KB

Name	Date modified	Type	Size
1	6/6/2017 10:08 PM	File folder	
DEAD	6/6/2017 10:05 PM	File folder	
LIVE	6/6/2017 10:05 PM	File folder	
OW	6/6/2017 10:05 PM	File folder	
\$DROPSDISC1\$	6/6/2017 10:05 PM	File	1 KB
1.B	6/6/2017 10:05 PM	B File	6 KB
1.drp	6/6/2017 10:05 PM	DRP File	1 KB
1.drp.prop	6/6/2017 10:05 PM	PROP File	1 KB
1.GEN	6/6/2017 10:09 PM	Structural Model	13 KB
1.LC	6/6/2017 10:05 PM	PLPost file	1 KB

6.1. File needed to be exported before using PTPAK

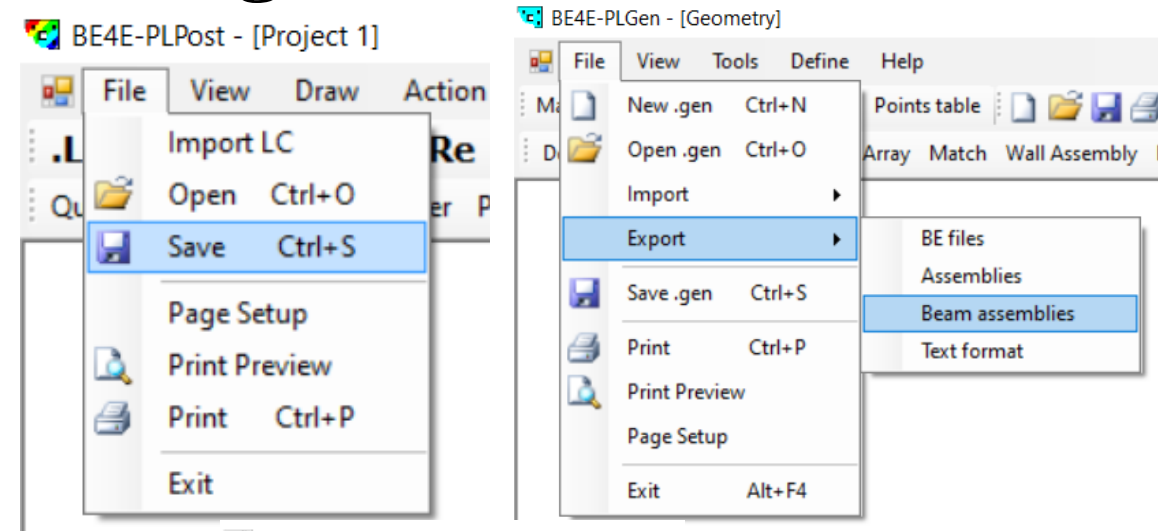
Same as PLDesign, there are cases that user have to export file from PLGen before using PTPAK:

There are cases that user have to export file from PLGen and PLPost before using PLDesign:

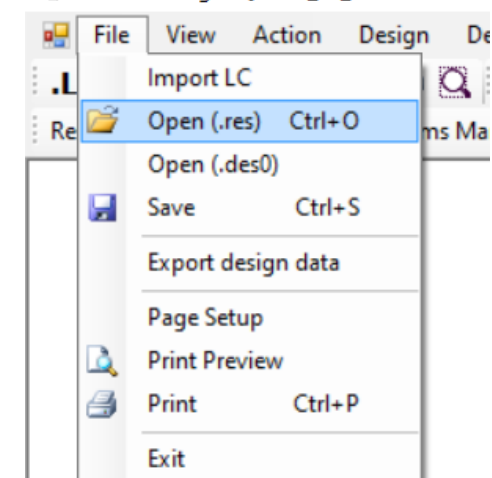
- Export beam assemblies: this case is used to design the beams.
- Save the PLPost results: this case is used to design the slab.
- Export assemblies file: this case is used to check punching of the columns.

The previous cases can be restored in the PLDesign as follows:

- Import beam assemblies: this case is used to design the beams (will be demonstrated in beam design section).
- Open the PLPost results: this case is used to design the slab.
- Import assemblies file: this case is used to check punching of the columns.

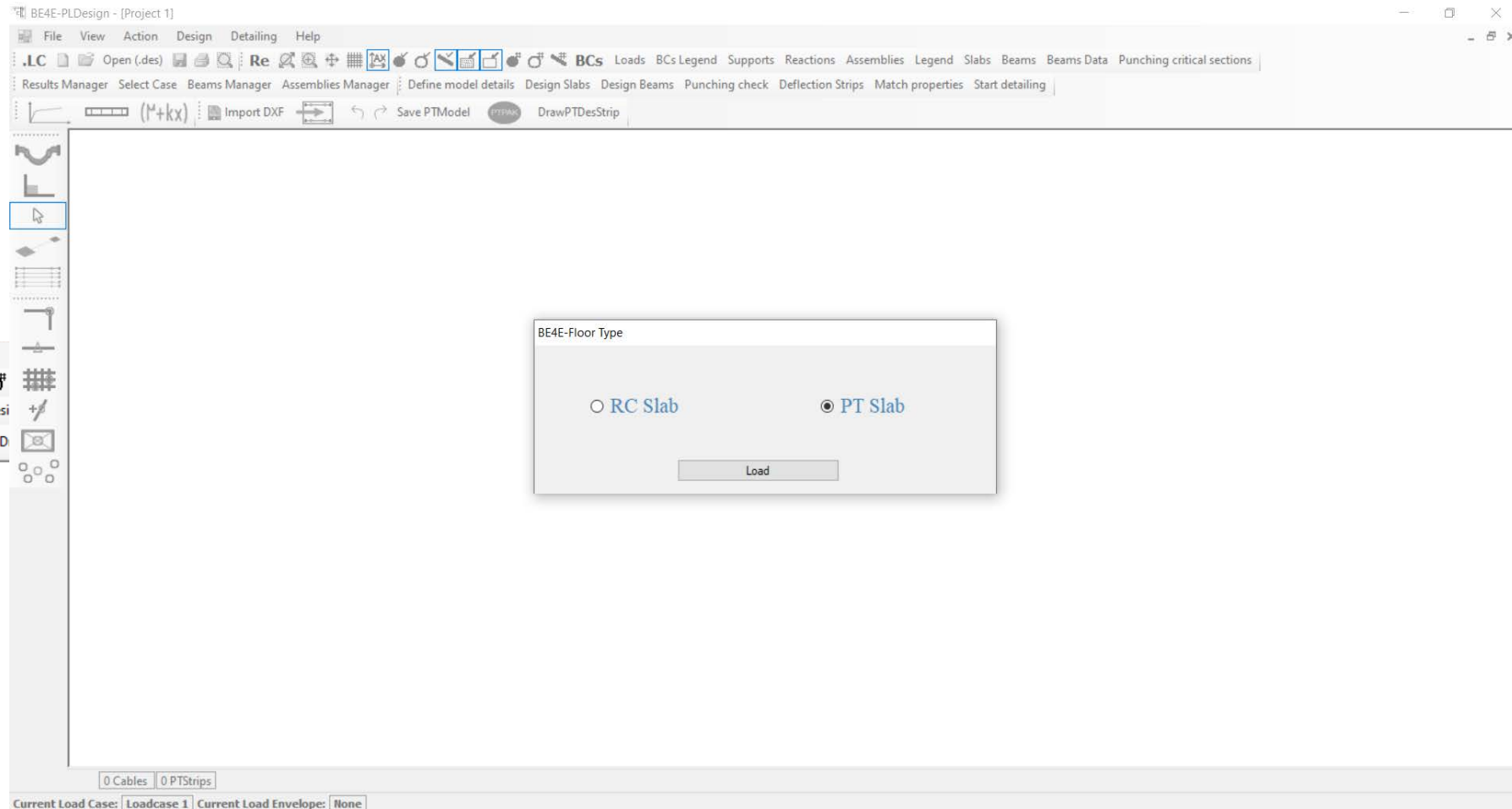


BE4E-PLDesign - [Floor_ID_300420.DL1-t



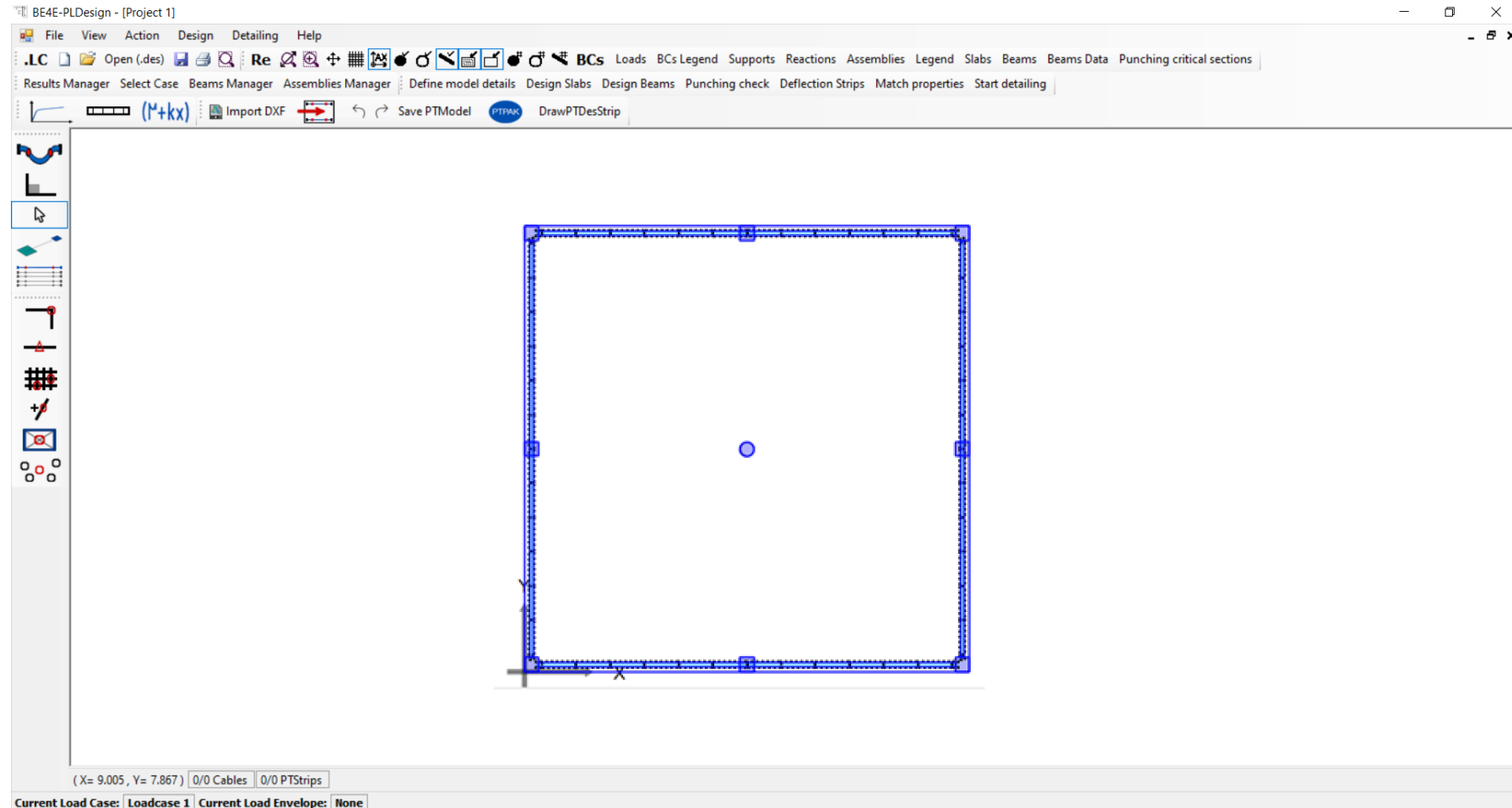
6.2 How can users generate PTPAK Model

- Choose PT Slab, hence choose Load
- Import Input File (.LC File).



6.2 How can users generate PTPAK Model

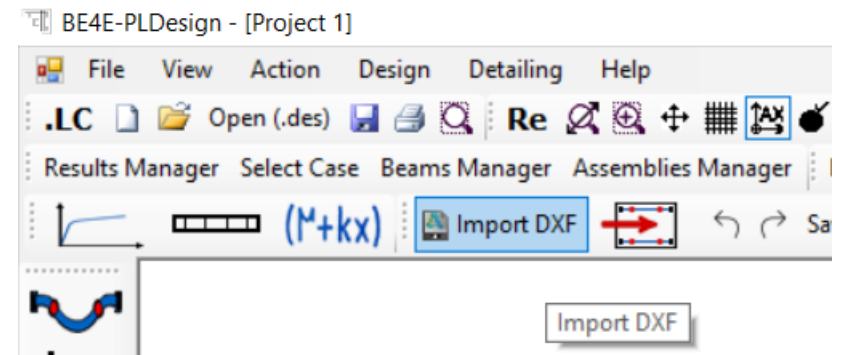
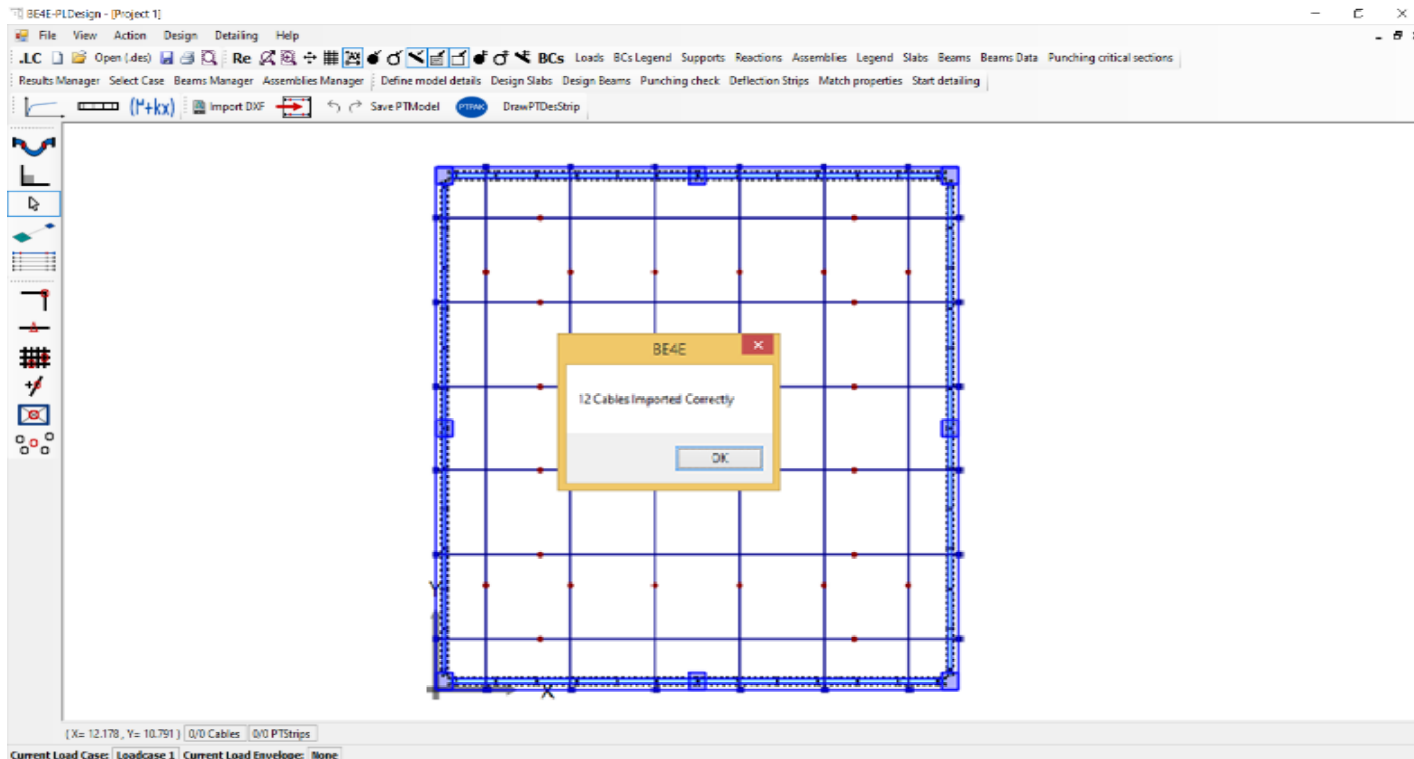
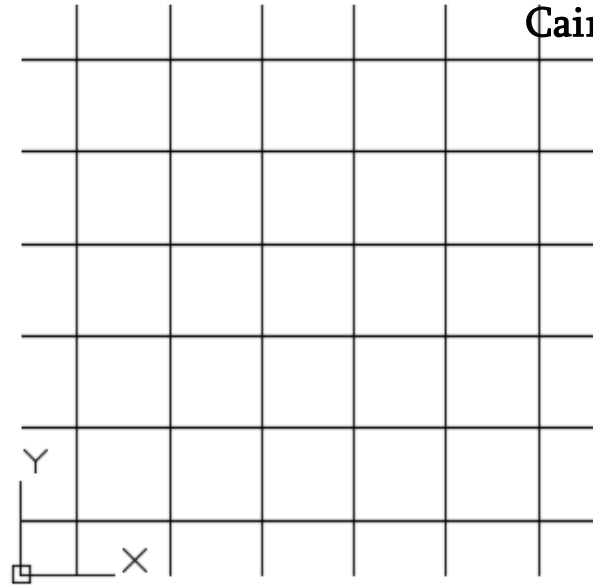
- There are two ways to draw cables:
 - 1- Export .Dxf for cables.
 - 2- Draw directly with snapping tools.



6.2 How can users generate PTPAK Model

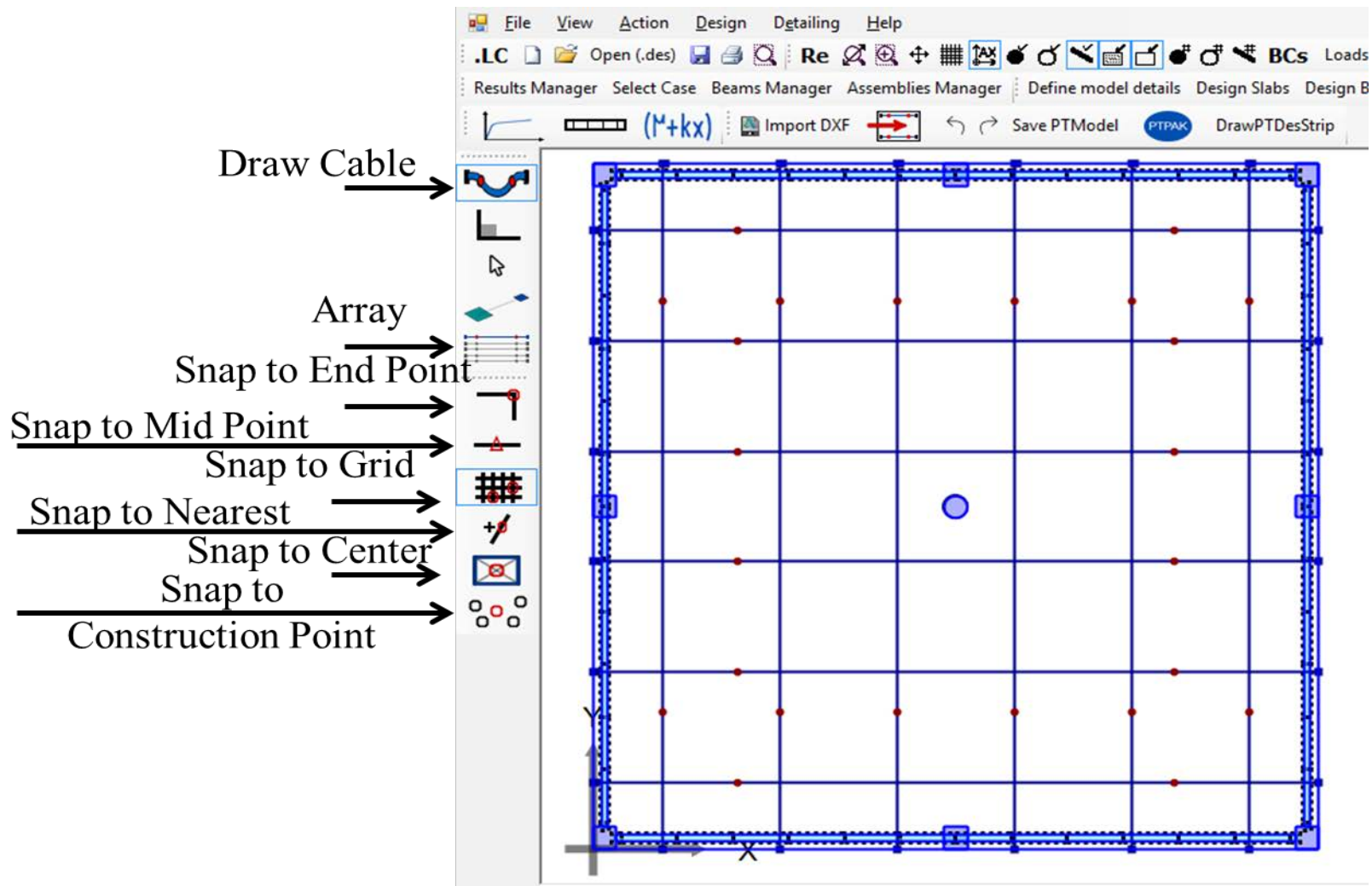
A - Export cables from .DXF file:

- Draw cables in AutoCAD file (Polyline) .
- Save as .DXF file.
- Import it from PTPAK.



6.2 How can users generate PTPAK Model

B- Draw Cables from Snap Tools:



The image shows a software interface for drawing cables. On the left, a vertical toolbar contains several snap tools, each with an arrow pointing to it from a text label:

- Draw Cable
- Array
- Snap to End Point
- Snap to Mid Point
- Snap to Grid
- Snap to Nearest
- Snap to Center
- Snap to
- Construction Point

The main window displays a grid of blue lines representing a slab. A blue circle is placed in the center of the grid. The software interface includes a menu bar (File, View, Action, Design, Detailing, Help) and a toolbar with various icons. The status bar at the bottom shows options like 'Save PTModel', 'PTPAK', and 'DrawPTDesStrip'.

6.2 How can users generate PTPAK Model

○ Define Cable Properties:



Prestressing Material ✕

Material Name

Material Properties

Modulus of Elasticity, Eps kN/m²

Ultimate Stress, Fpu kN/m²

Yield Stress, Fpy kN/m²

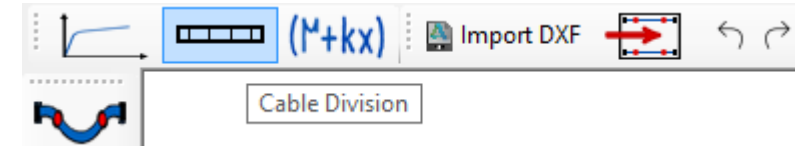
Area Strands, Aps m²

Code Provisions

Maximum Allowable Stress by jacking, Fpi

User Defiend

○ Define Cable divisions:



Cable Division — □ ✕

Number of Divisions

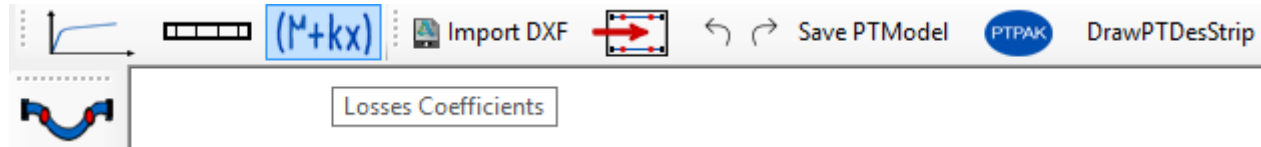
Cable Width m

Number of Divisions

Aspect Ratio

6.2 How can users generate PTPAK Model

- Define Losses Coefficients:



Losses Data

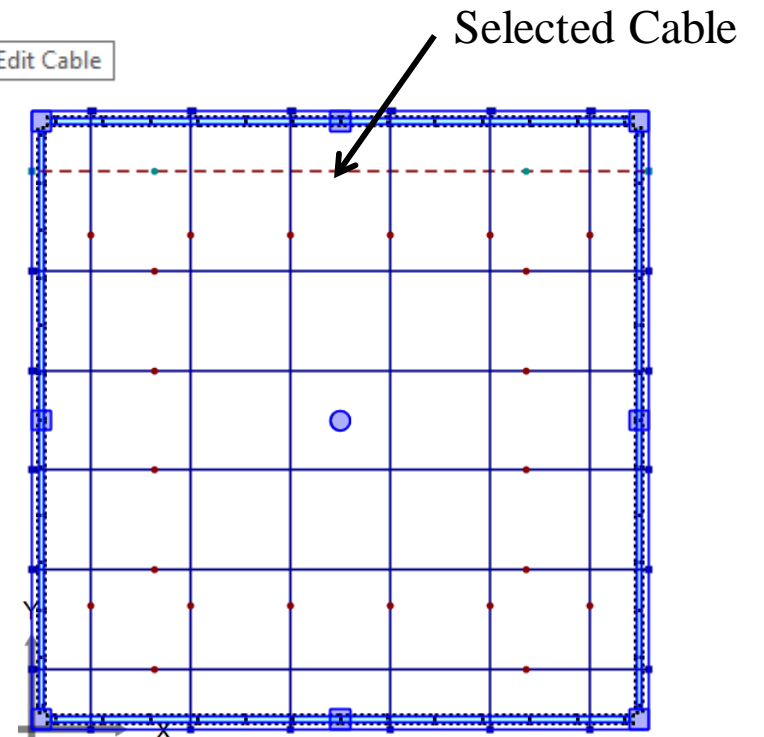
Coefficients

Friction Coefficient, M

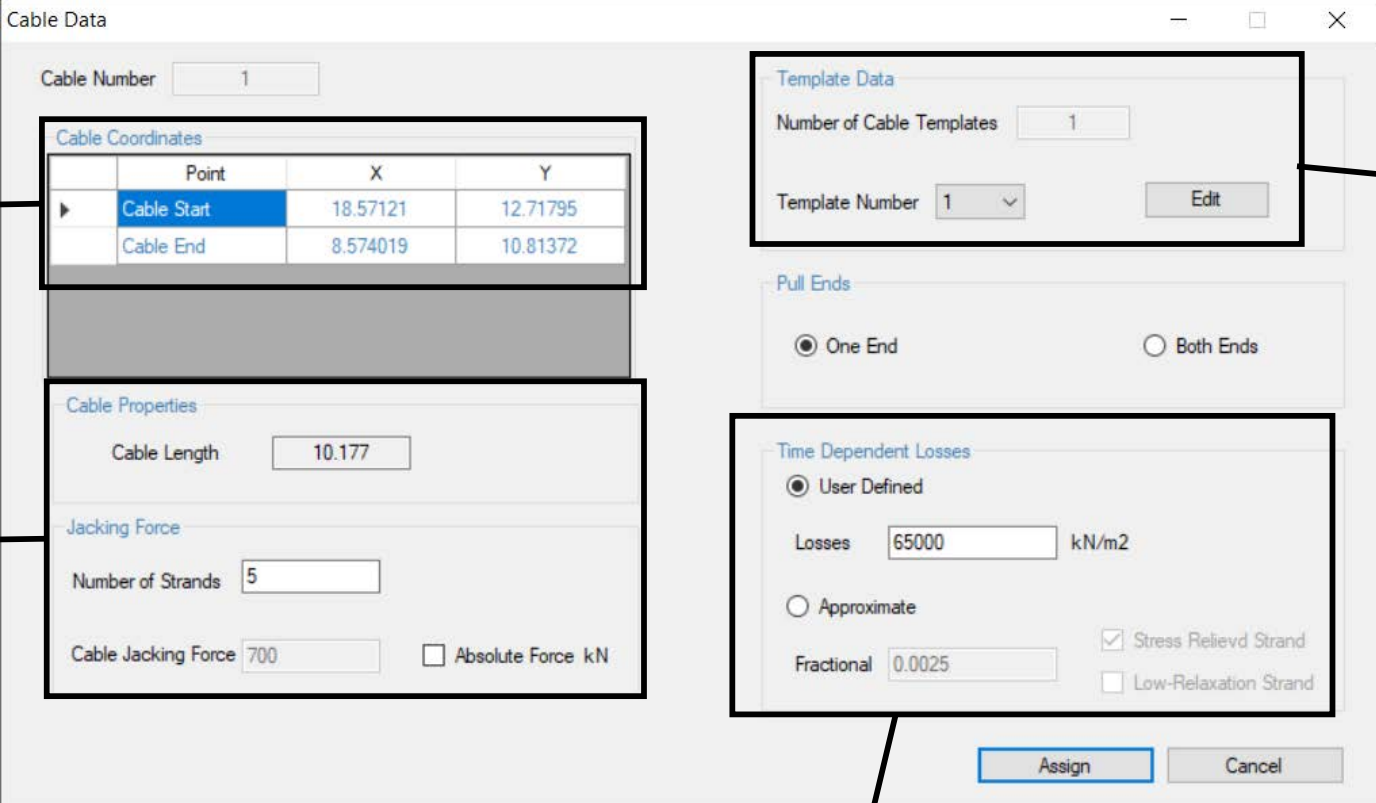
Wobble Coefficient, W

Seating Losses mm

- Define the Cable Profile:
 - Select the cable/cables.
 - Then right click to open the cable data.



6.2 How can users generate PTPAK Model



The screenshot shows the 'Cable Data' dialog box with several sections highlighted by black boxes and annotated with arrows:

- Cable (Start/End) Coordinates:** Points to the 'Cable Coordinates' table.
- Cable Length/ Jacking Force:** Points to the 'Cable Properties' and 'Jacking Force' sections.
- Cable Template (13 different template):** Points to the 'Template Data' section.
- Time dependent Losses:** Points to the 'Time Dependent Losses' section.

Point	X	Y
Cable Start	18.57121	12.71795
Cable End	8.574019	10.81372

Cable Properties: Cable Length: 10.177

Jacking Force: Number of Strands: 5, Cable Jacking Force: 700, Absolute Force kN

Template Data: Number of Cable Templates: 1, Template Number: 1, Edit button

Pull Ends: One End, Both Ends

Time Dependent Losses: User Defined, Losses: 65000 kN/m², Approximate, Fractional: 0.0025, Stress Relievd Strand, Low-Relaxation Strand

Buttons: Assign, Cancel

6.2 How can users generate PTPAK Model

- Click on edit to edit the template data.

Template Data

Number of Cable Templates

Template Number

Template Properties

Template Number

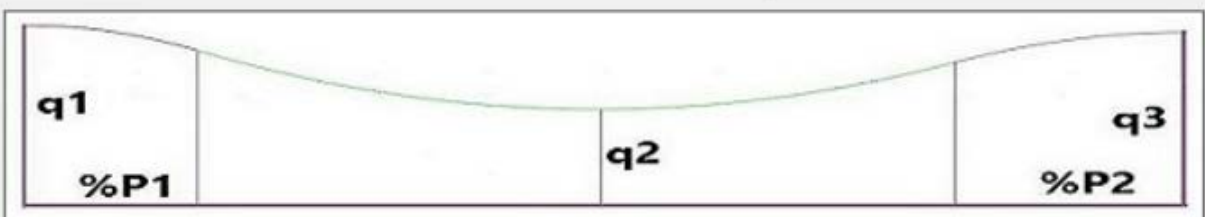
Template Type

Template Coordinates

Point	X	Y
Template Start	18.57121	12.71795
Template End	8.574019	10.81372

Template Properties

Template Length



Template Parameters

q1 P1 * Percentage of Length

q2 P2

q3

NOTE: q1 not equal q3

6.2 How can users generate PTPAK Model

Template Properties


Template Number

Template Type

Template Coordinates			
	Point	X	Y
▶	Template Start	18.57121	12.71795
	Template End	8.574019	10.81372

Template Properties

Template Length



Template Parameters

q1 P1 * Percentage of Length

q2

Template 2

Template Properties


Template Number

Template Type

Template Coordinates			
	Point	X	Y
▶	Template Start	18.57121	12.71795
	Template End	8.574019	10.81372

Template Properties

Template Length



Template Parameters

q1 P1 * Percentage of Length

q2

Template 3

6.2 How can users generate PTPAK Model

Template Properties

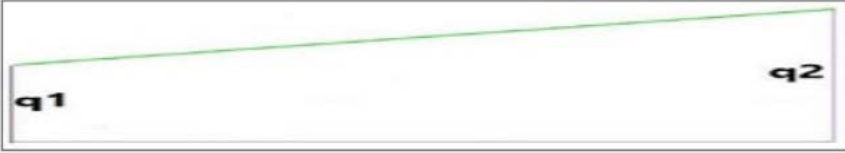
Template Number

Template Type

Template Coordinates			
	Point	X	Y
▶	Template Start	18.57121	12.71795
	Template End	8.574019	10.81372

Template Properties

Template Length



Template Parameters

q1 * Percentage of Length

q2

Template 4

Template Properties


Template Number

Template Type

Template Coordinates			
	Point	X	Y
▶	Template Start	18.57121	12.71795
	Template End	8.574019	10.81372

Template Properties

Template Length



Template Parameters

q1 * Percentage of Length

q2

q3

NOTE: $2 \times q3 > q1 + q2$

Template 5

6.2 How can users generate PTPAK Model

Template Properties

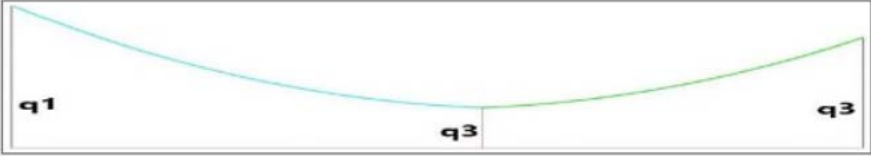
Template Number

Template Type

Template Coordinates			
	Point	X	Y
▶	Template Start	18.57121	12.71795
	Template End	8.574019	10.81372

Template Properties

Template Length



Template Parameters

q1 * Percentage of Length

q2

q3

NOTE: $2 \times q3 > q1 + q2$

Template 6

Template Properties

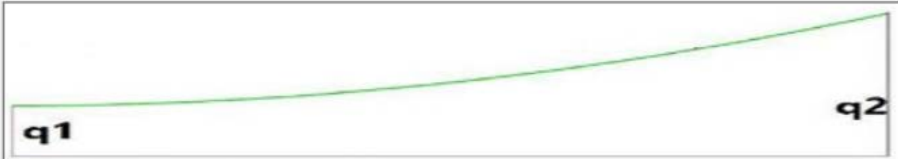
Template Number

Template Type

Template Coordinates			
	Point	X	Y
▶	Template Start	18.57121	12.71795
	Template End	8.574019	10.81372

Template Properties

Template Length



Template Parameters

q1 * Percentage of Length

q2

Template 7

6.2 How can users generate PTPAK Model

Template Properties

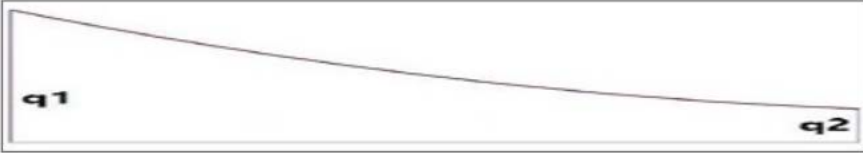
Template Number

Template Type

Template Coordinates			
	Point	X	Y
▶	Template Start	18.57121	12.71795
	Template End	8.574019	10.81372

Template Properties

Template Length



Template Parameters

q1 * Percentage of Length

q2

Template 8

Template Properties


Template Number

Template Type

Template Coordinates			
	Point	X	Y
▶	Template Start	18.57121	12.71795
	Template End	8.574019	10.81372

Template Properties

Template Length



Template Parameters

q1 * Percentage of Length

q2

Template 9

6.2 How can users generate PTPAK Model

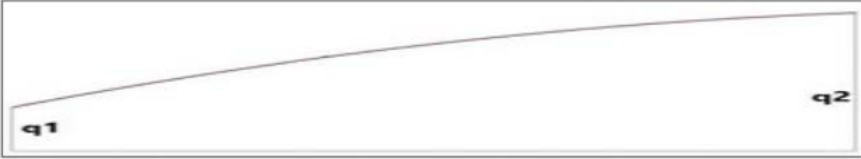
Template Properties

Template Number: 1

Template Type: Template_10

Point	X	Y
Template Start	18.57121	12.71795
Template End	8.574019	10.81372

Template Length: 10.177



Template Parameters

q1: 0.27 * Percentage of Length

q2: 0.03

Assign

Template 10

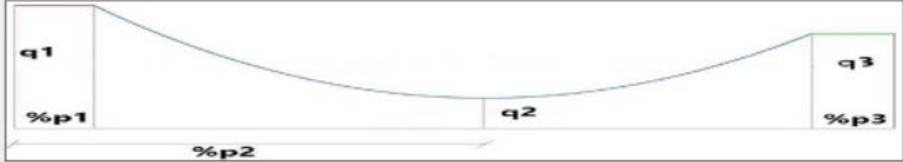
Template Properties

Template Number: 1

Template Type: Template_11

Point	X	Y
Template Start	18.57121	12.71795
Template End	8.574019	10.81372

Template Length: 10.177



Template Parameters

q1: 0.27 * Percentage of Length

q2: 0.03

q3: 0.265

P1: 0.2 * Percentage of Length

P2: 0.2

P3: 0.2

Assign

Template 11

6.2 How can users generate PTPAK Model

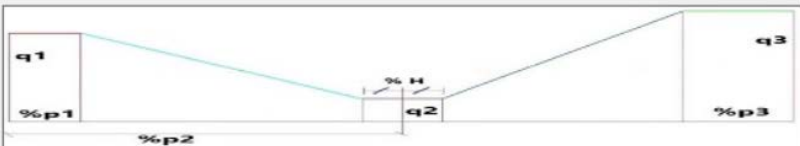
Template Properties

Template Number

Template Type

Point	X	Y
▶ Template Start	18.57121	12.71795
Template End	8.574019	10.81372

Template Length



Template Parameters

q1 P1 * Percentage of Length

q2 P2

q3 P3

H

Template 12

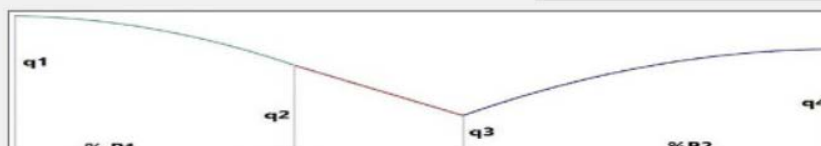
Template Properties

Template Number

Template Type

Point	X	Y
▶ Template Start	18.57121	12.71795
Template End	8.574019	10.81372

Template Length



Template Parameters

q1 P1 * Percentage of Length

q2 P2

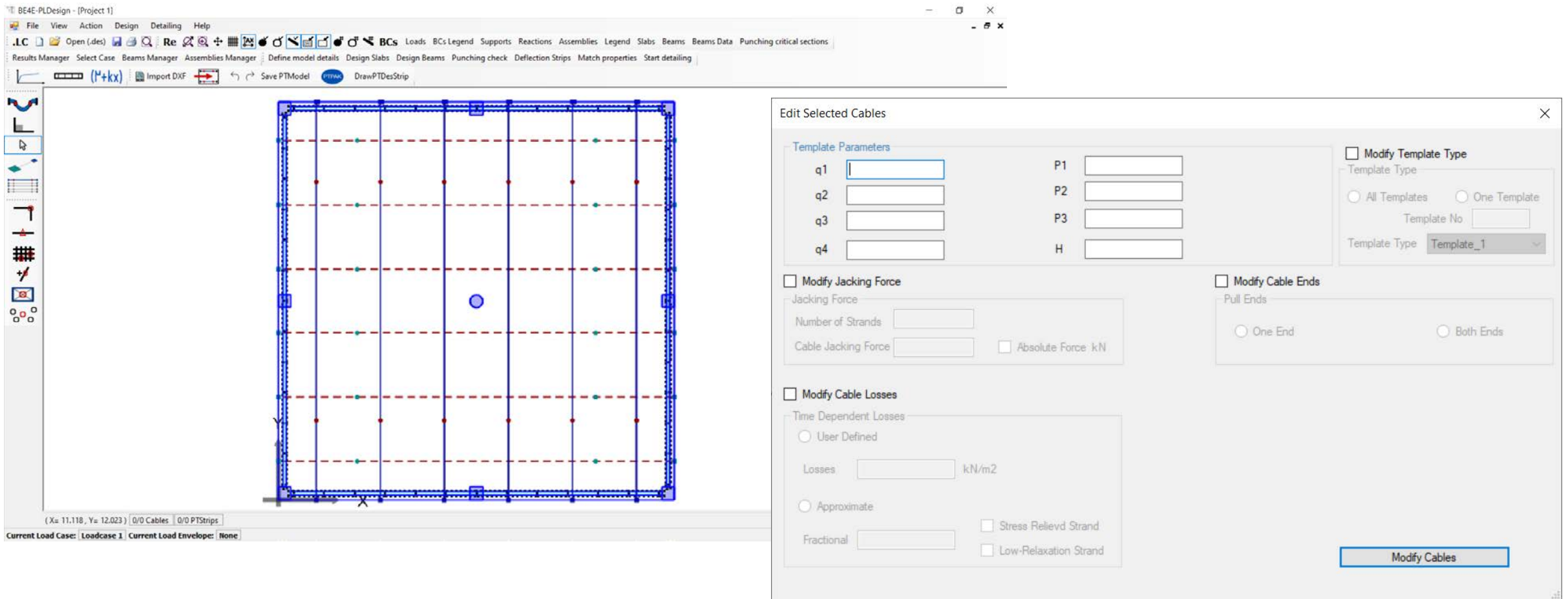
q3

q4

Template 13

6.2 How can users generate PTPAK Model

Instead of choosing cable properties one by one, the user can choose multiple cables with the same properties.



The screenshot displays the BE4E-PLDesign software interface. The main window shows a grid of cables (blue lines) within a rectangular boundary (dashed red lines). The 'Edit Selected Cables' dialog box is open, allowing users to modify cable properties. The dialog includes the following sections:

- Template Parameters:** Fields for q1, q2, q3, q4, P1, P2, P3, and H.
- Modify Template Type:** Radio buttons for 'All Templates' and 'One Template', a 'Template No' field, and a 'Template Type' dropdown menu.
- Modify Jacking Force:** A checkbox to enable this section, with fields for 'Jacking Force', 'Number of Strands', 'Cable Jacking Force', and an 'Absolute Force: kN' checkbox.
- Modify Cable Ends:** A checkbox to enable this section, with a 'Pull Ends' section containing radio buttons for 'One End' and 'Both Ends'.
- Modify Cable Losses:** A checkbox to enable this section, with radio buttons for 'User Defined' and 'Approximate'. The 'User Defined' option includes a 'Losses' field (kN/m²). The 'Approximate' option includes a 'Fractional' field and checkboxes for 'Stress Relievd Strand' and 'Low-Relaxation Strand'.

A 'Modify Cables' button is located at the bottom right of the dialog box. The software interface also shows a menu bar (File, View, Action, Design, Detailing, Help) and a toolbar with various icons. The status bar at the bottom indicates '(X= 11.118, Y= 12.023) | 0/0 Cables | 0/0 PTPStrips' and 'Current Load Case: Loadcase 1 | Current Load Envelope: None'.

6. PTPAK Package

6.1 Files need to be exported before using PTPAK

6.2 How can users generate PTPAK Model

6.2.1 Draw cables ✓

6.2.2 Cables data ✓

6.2.3 Cable templates ✓

6.2.4 Multiple cable selection ✓

6.3 How to solve the PTPAK model

6.4 Load combinations & load envelopes

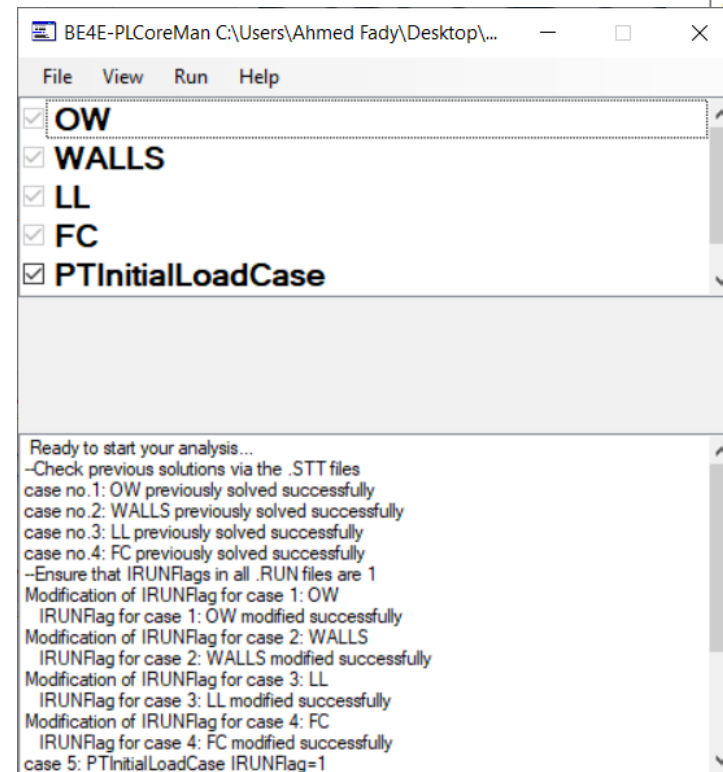
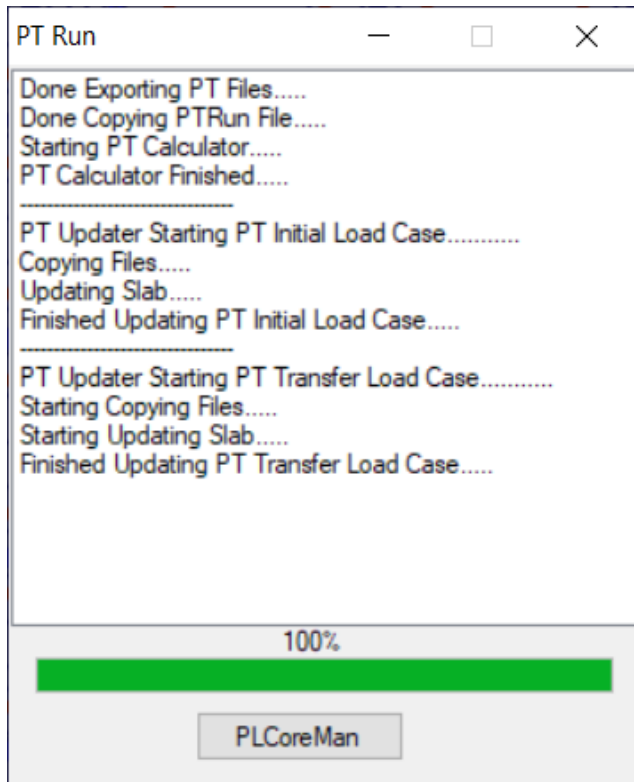
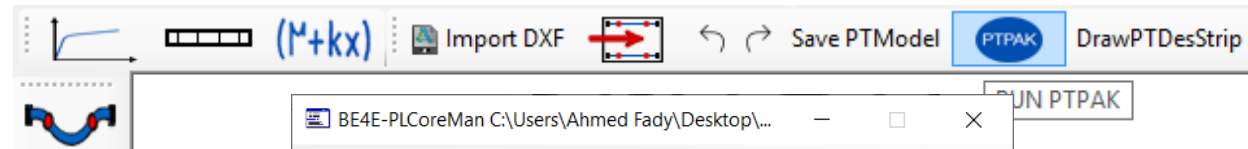
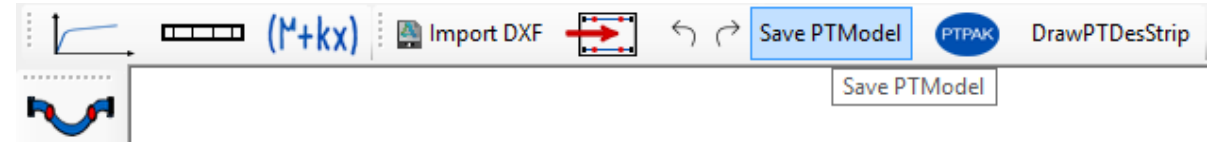
6.5 Check the Cable Eccentricity

6.6 Check the Cable stresses

6.7 Optimizer

6.3 How to solve the PTPAK model

- Before solving PTPAK, the user should save the PTModel.
- Hence, run PTPAK.
- Open PLCoreMan, and run PL.exe.



6. PTPAK Package

6.1 Files need to be exported before using PTPAK

6.2 How can users generate PTPAK Model

6.2.1 Draw cables ✓

6.2.2 Cables data ✓

6.2.3 Cable templates ✓

6.2.4 Multiple cable selection ✓

6.3 How to solve the PTPAK model ✓

6.4 Load combinations & load envelopes

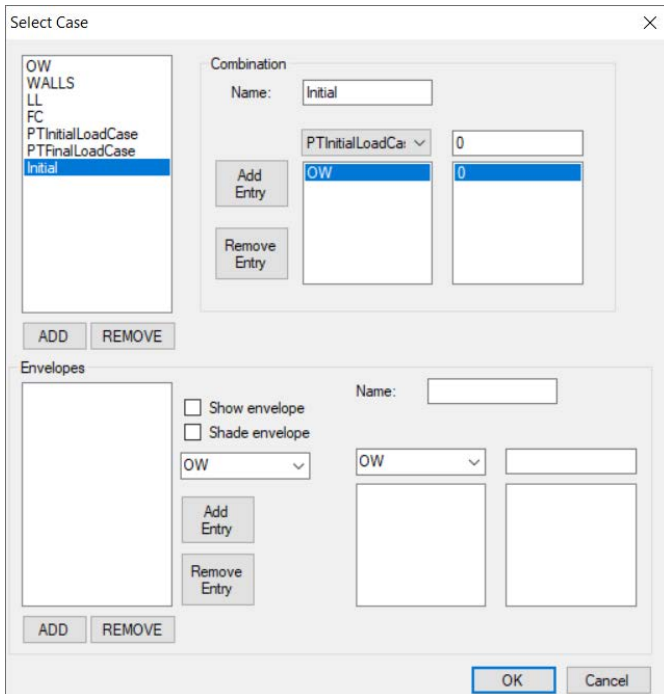
6.5 Check the Cable Eccentricity

6.6 Check the Cable stresses

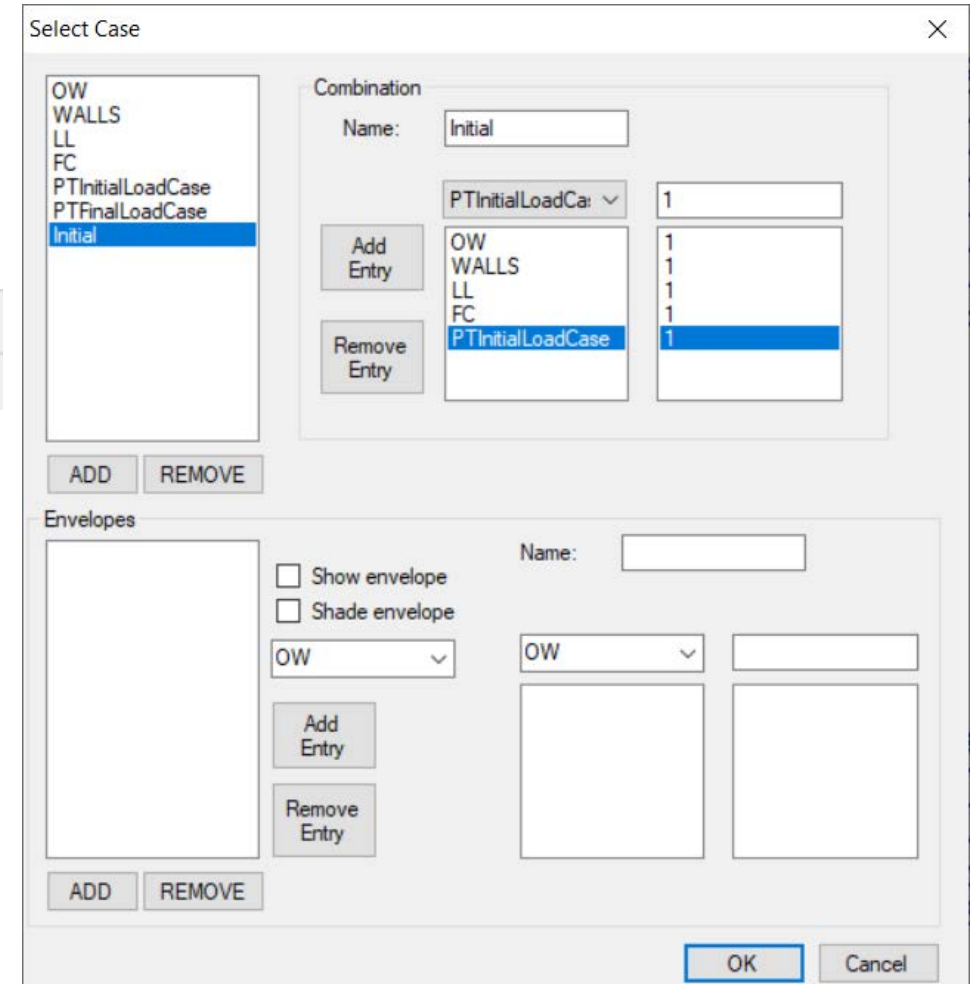
6.7 Optimizer

6.4 Load combinations & load envelopes

- Reopen the PTPAK.exe to apply load combinations.
- The lower tabs of the PTPAK, If the user press double click on OW Combinations window will open.

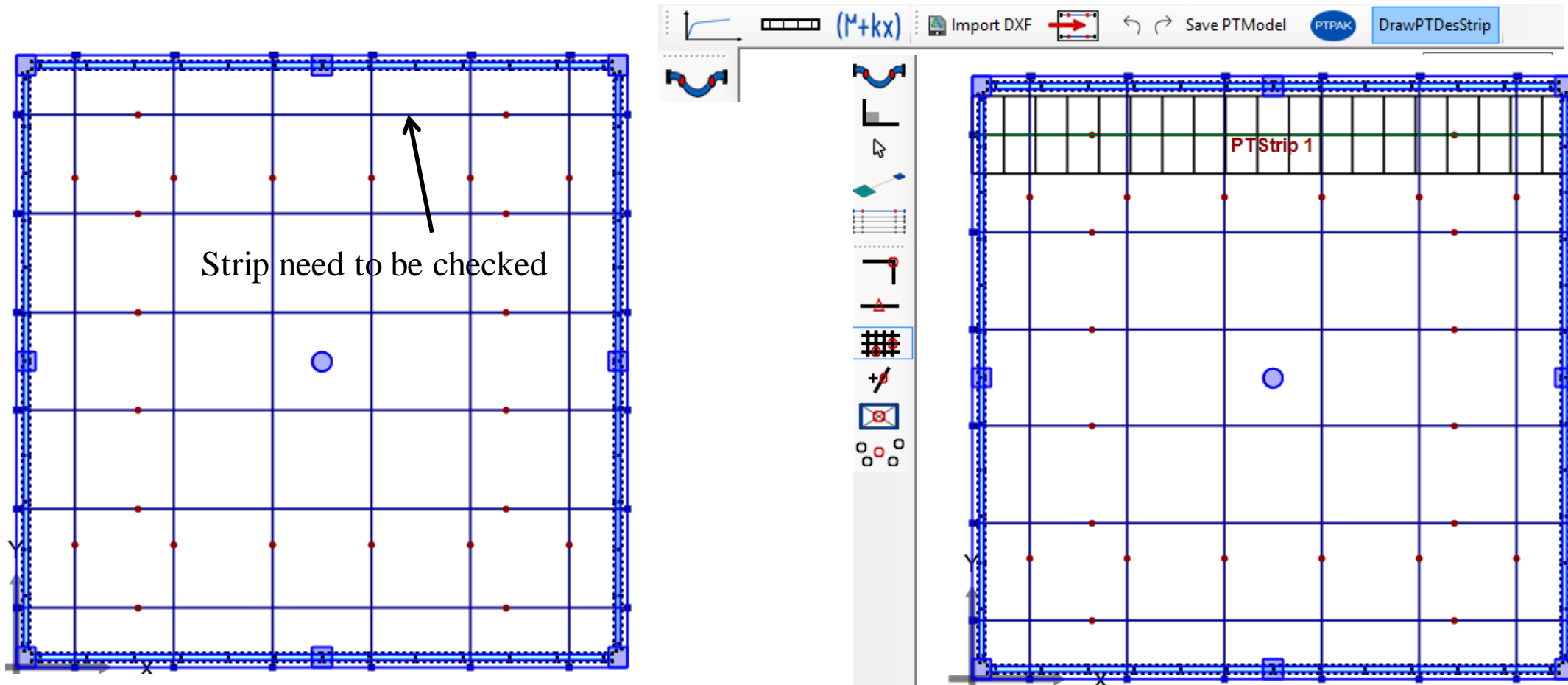


(X= -13.028 , Y= 14.413) 0/0 Cables 0/0 PTStrips
Load Case: OW Current Load Envelope: None



6.5 Check the Cable Eccentricity

- The user should check the eccentricity of the cable profile and the stress distribution through the slab cross section.



6.5 Check the Cable Eccentricity

- User should draw the design strip and edit its prosperities.

PT Strip

Strip Number:

Number of sections at each strip

Strip Properties

Strip Width: Sections:

Design Parameters

Bending Moment: Load Cases: Load Combinations:

Section Area: m²

Section Modulus: m³

Strip Coordinates

Point	X	Y
Strip Start	3.802193	27.28387
Strip End	25.54053	26.68557

Export Strip Sections

Monitoring Button

Stresses

10¹

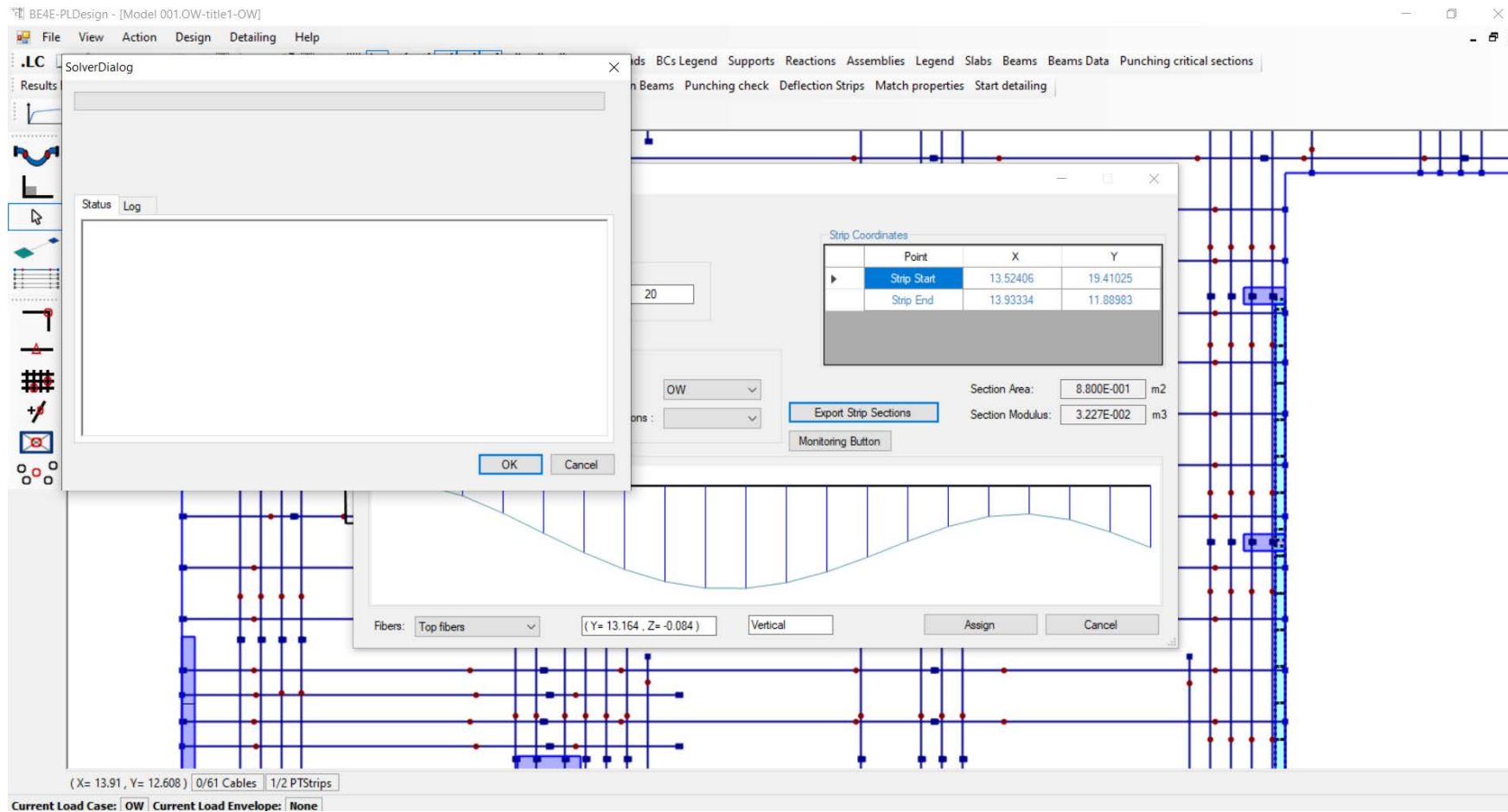
Fibers: Horizontal

The direction of bending moment to calculate the section stresses

Load Cases and Load Combination

6.6 Check the Cable stresses

- User export strip sections.



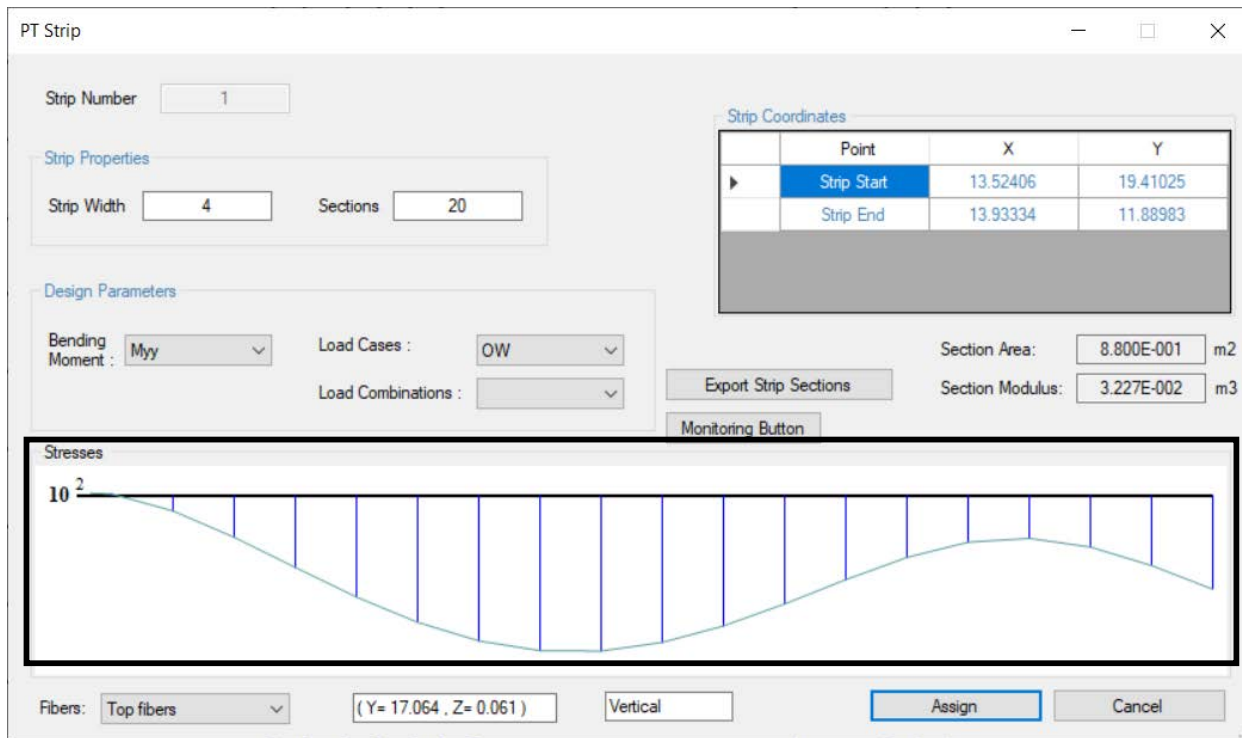
The screenshot displays the BE4E-PLDesign software interface. The main window shows a grid of cables with a deflection curve overlaid. A 'Strip Coordinates' dialog box is open, displaying the following data:

Point	X	Y
Strip Start	13.52406	19.41025
Strip End	13.93334	11.88983

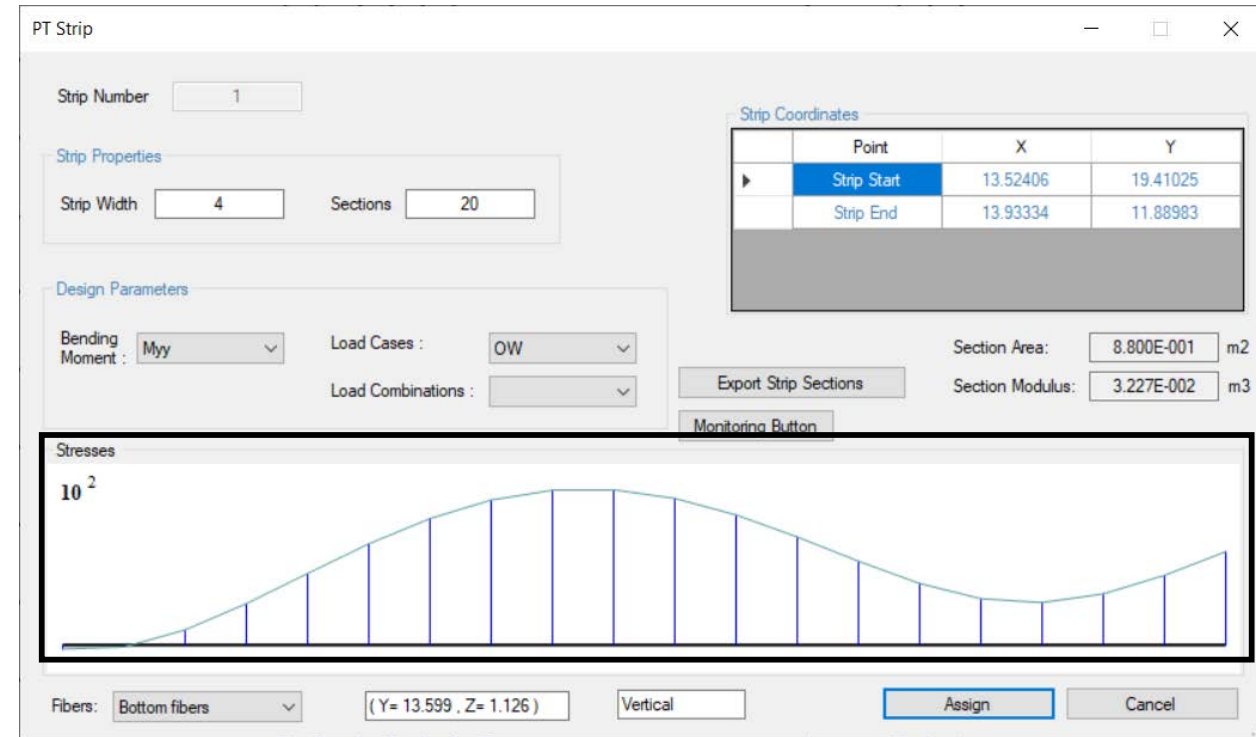
Below the table, the 'Section Area' is 8.800E-001 m² and the 'Section Modulus' is 3.227E-002 m³. The 'Export Strip Sections' button is highlighted. A 'Monitoring Button' is also visible. The status bar at the bottom indicates '(X= 13.91 , Y= 12.608) | 0/61 Cables | 1/2 PTStrips' and 'Current Load Case: OW | Current Load Envelope: None'.

6.6 Check the Cable stresses

- Hence, click on monitoring button to check the stresses.



Top stress distribution over the PTDesign Strip



Bottom stress distribution over the PTDesign Strip

6. PTPAK Package

6.1 Files need to be exported before using PTPAK

6.2 How can users generate PTPAK Model

6.2.1 Draw cables ✓

6.2.2 Cables data ✓

6.2.3 Cable templates ✓

6.2.4 Multiple cable selection ✓

6.3 How to solve the PTPAK model ✓

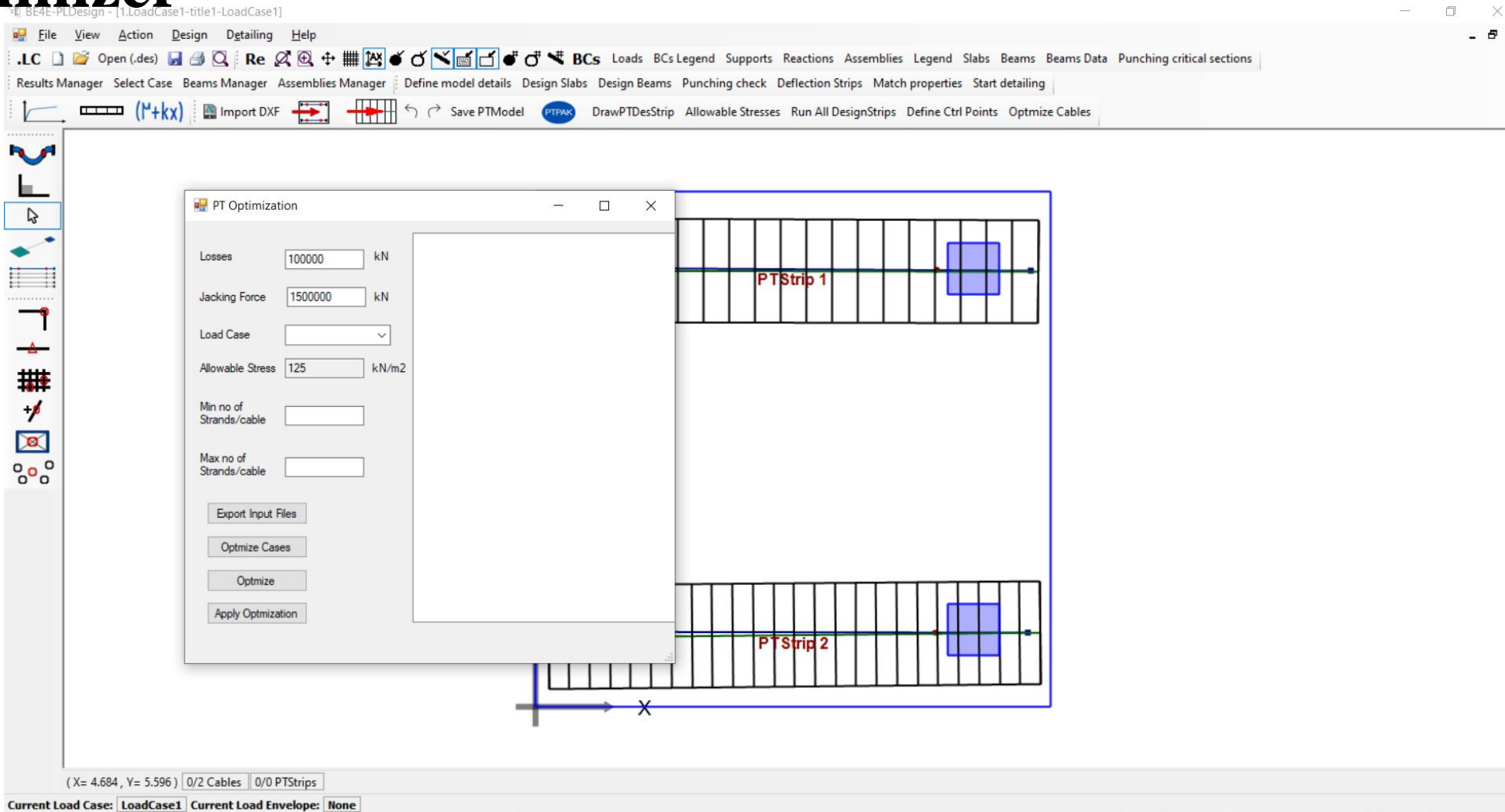
6.4 Load combinations & load envelopes ✓

6.5 Check the Cable Eccentricity ✓

6.6 Check the Cable stresses ✓

6.7 Optimizer

6.7 Optimizer

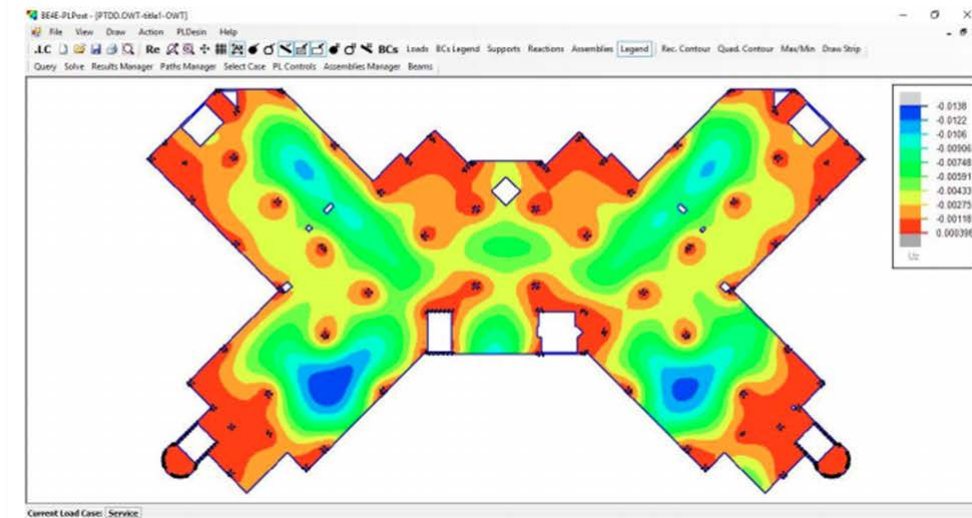
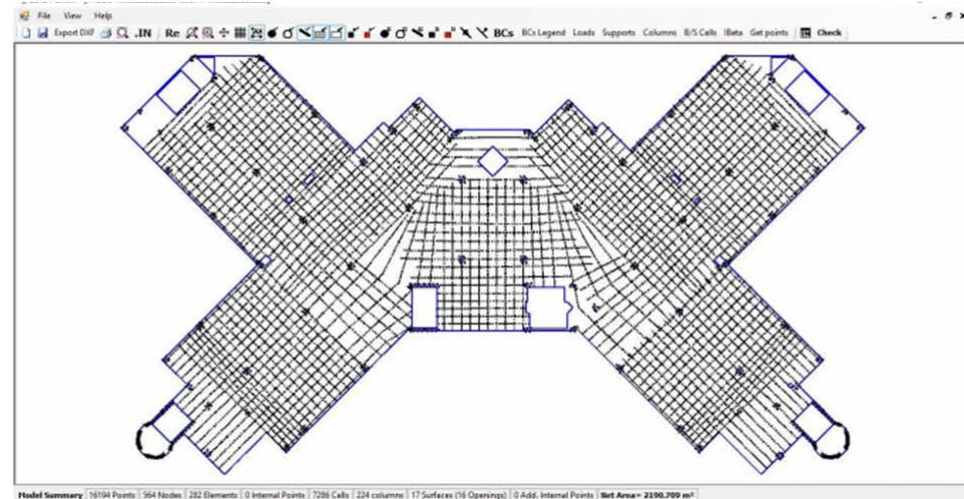
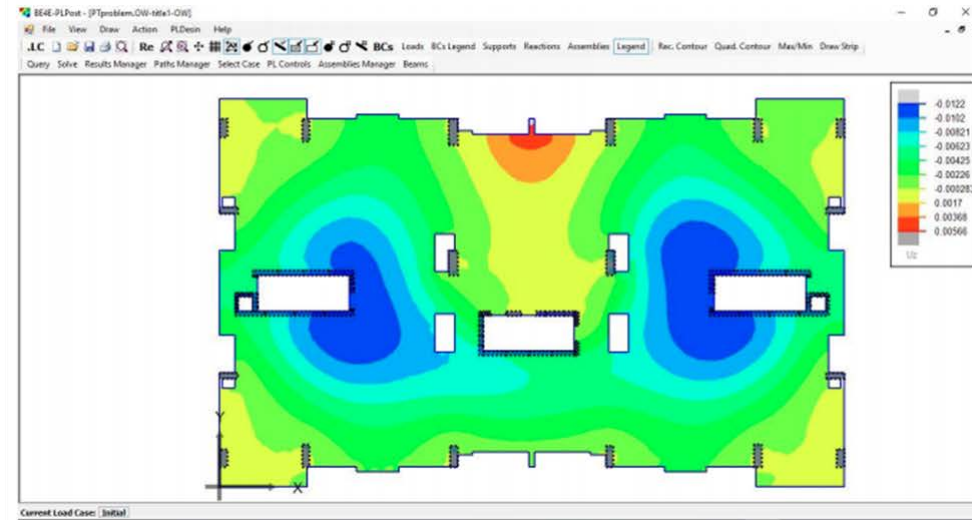
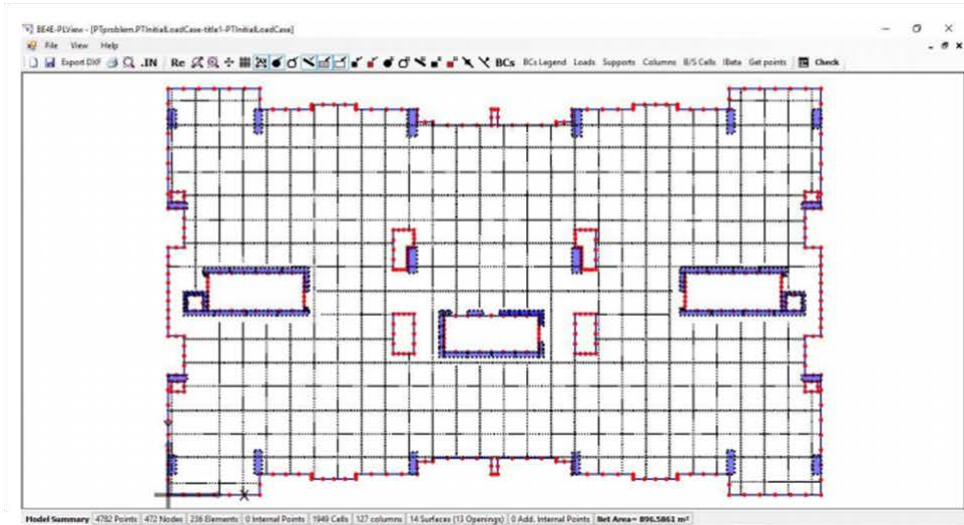


The screenshot displays the PT Optimization software interface. The main window shows a structural model with two horizontal beams, labeled "PT Strip 1" and "PT Strip 2", each with a blue square representing a cable. A "PT Optimization" dialog box is open, showing the following parameters:

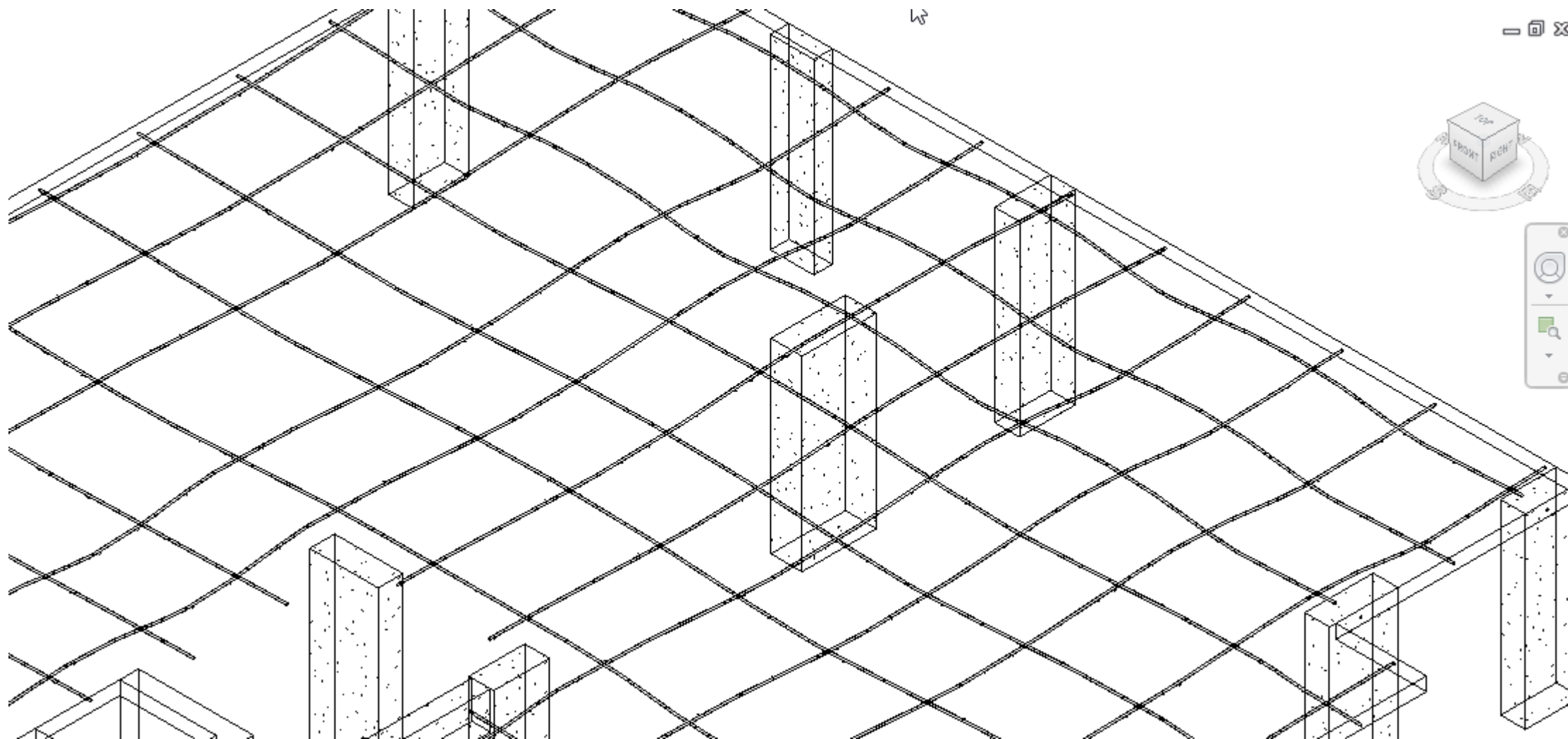
- Losses: 100000 kN
- Jacking Force: 1500000 kN
- Load Case: (dropdown menu)
- Allowable Stress: 125 kN/m²
- Min no of Strands/cable: (input field)
- Max no of Strands/cable: (input field)

Buttons in the dialog include "Export Input Files", "Optimize Cases", "Optimize", and "Apply Optimization". The status bar at the bottom indicates "(X= 4.684 , Y= 5.596) 0/2 Cables 0/0 PTStrips" and "Current Load Case: LoadCase1 Current Load Envelope: None".

6. Post-tension Package (PT) www.be4e.com



6. Post-tension Package (PT) www.be4e.com



CUFE-BE



Cairo University

6. Post-tension Package (PPT) www.be4e.com

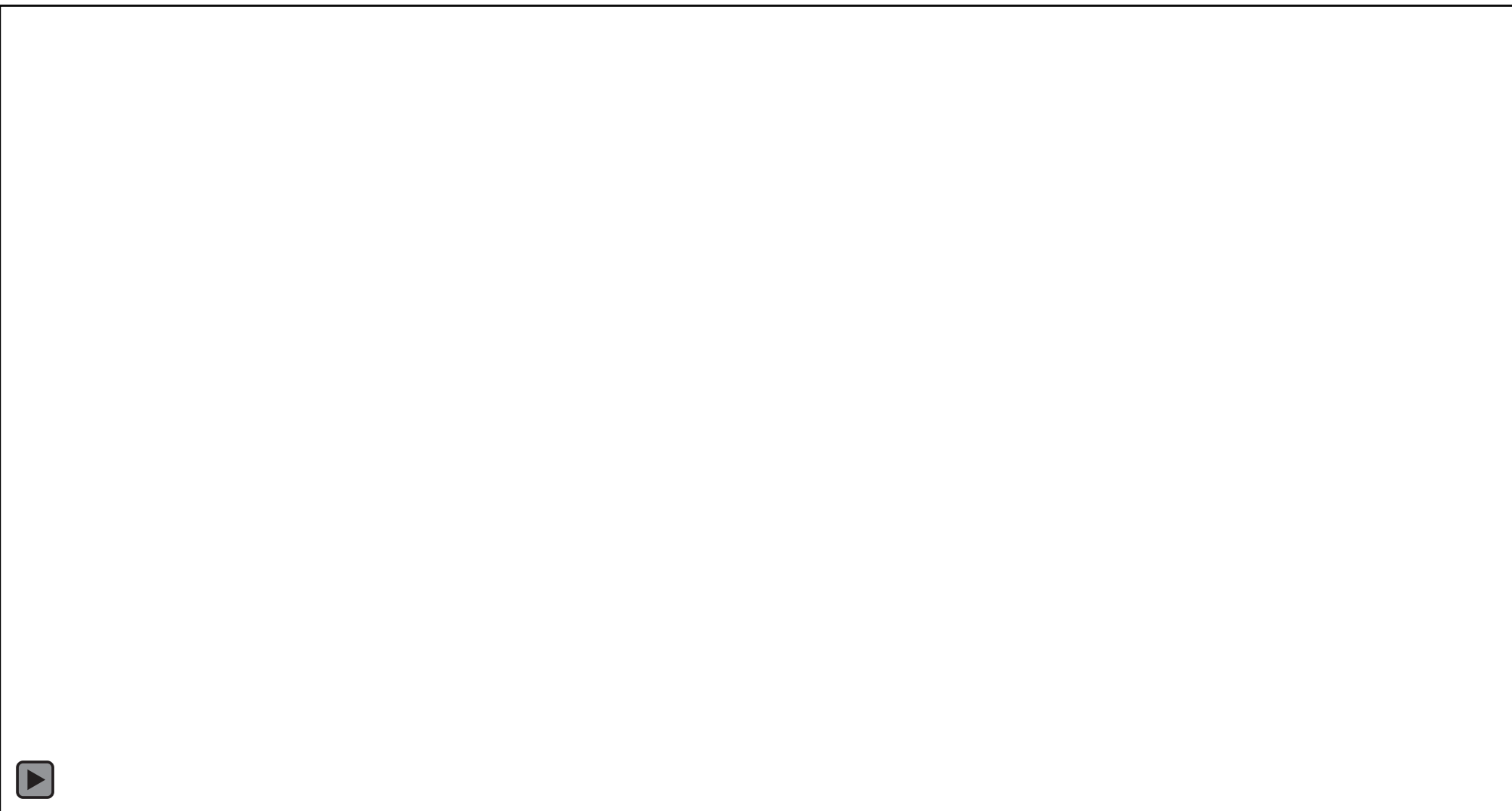
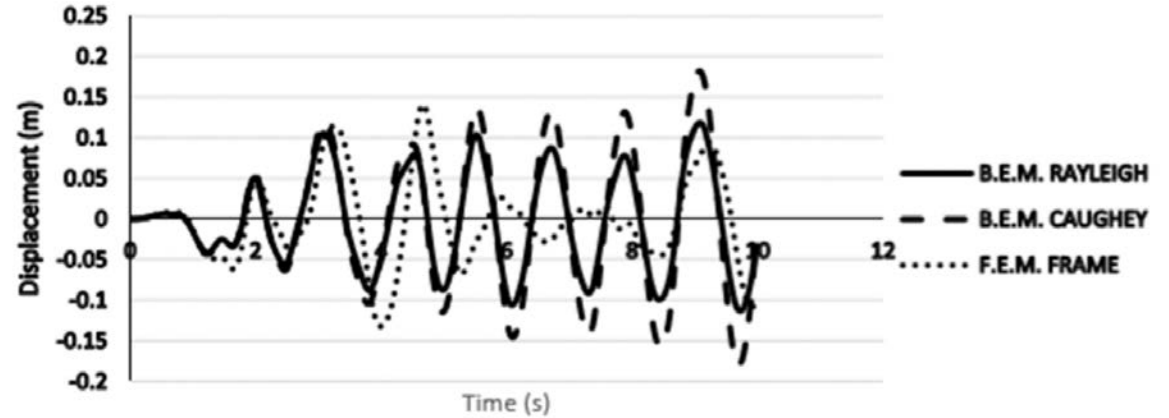


Table of contents

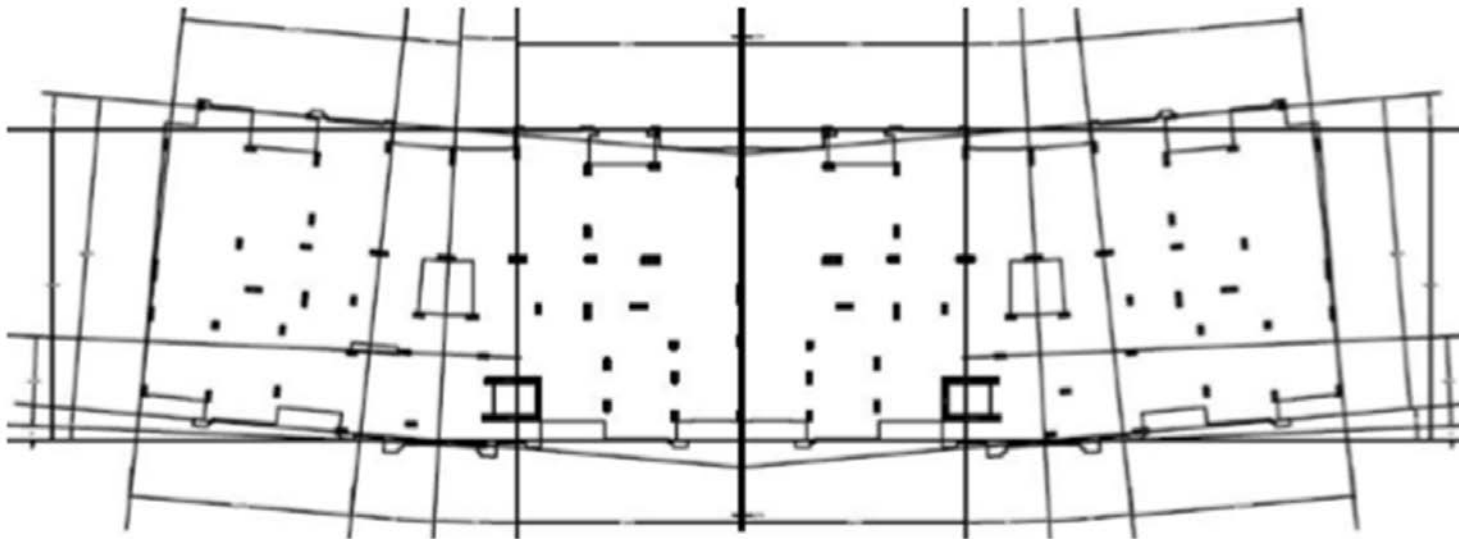
- 1. Introduction**
- 2. Basic package (PLPAK Basic)**
- 3. One floor package (BIM-PLPAK)**
- 4. Foundation Package (FoundPAK)**
- 5. Fixed base Package (FBPAK)**
- 6. Post-tension Package (PTPAK)**
- 7. Dynamic package (DynPAK)**
8. Overall building package (OBPAK)
9. 4D and 5D analysis
10. Conclusions

7. Dynamic package (DynPan)

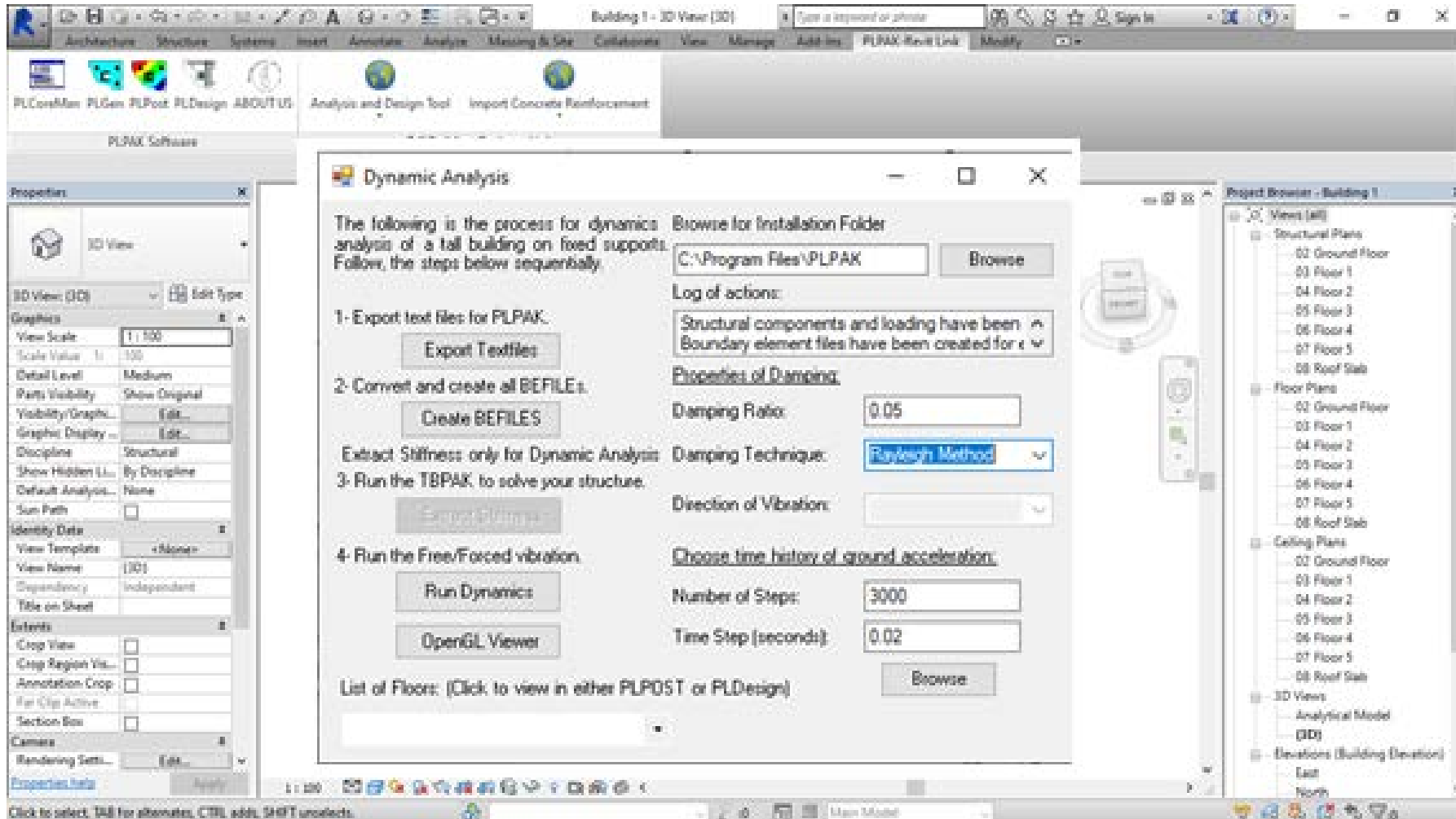
- Free vibration analysis
- Time history analysis
- Damping effect
- Performance based design



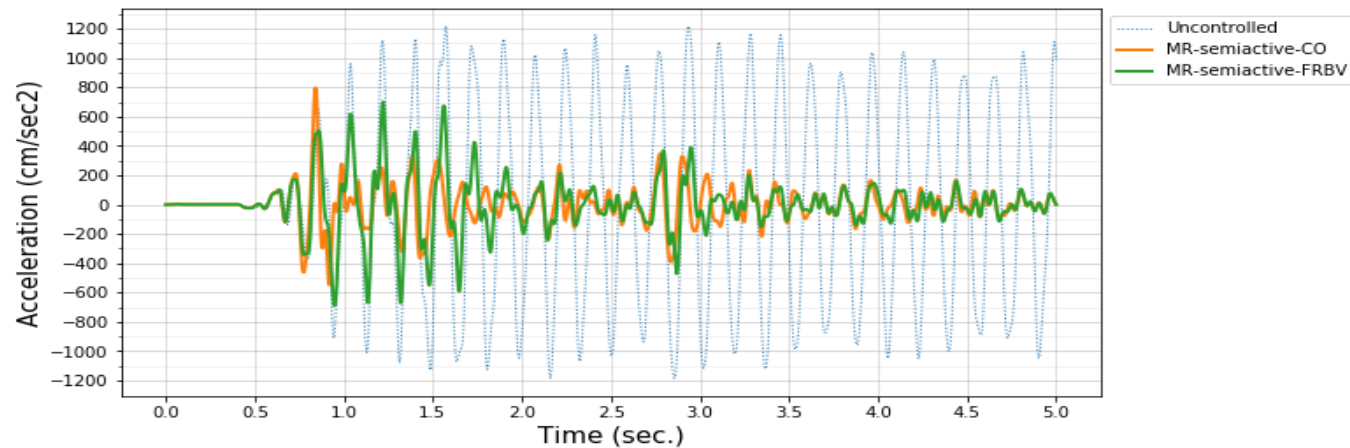
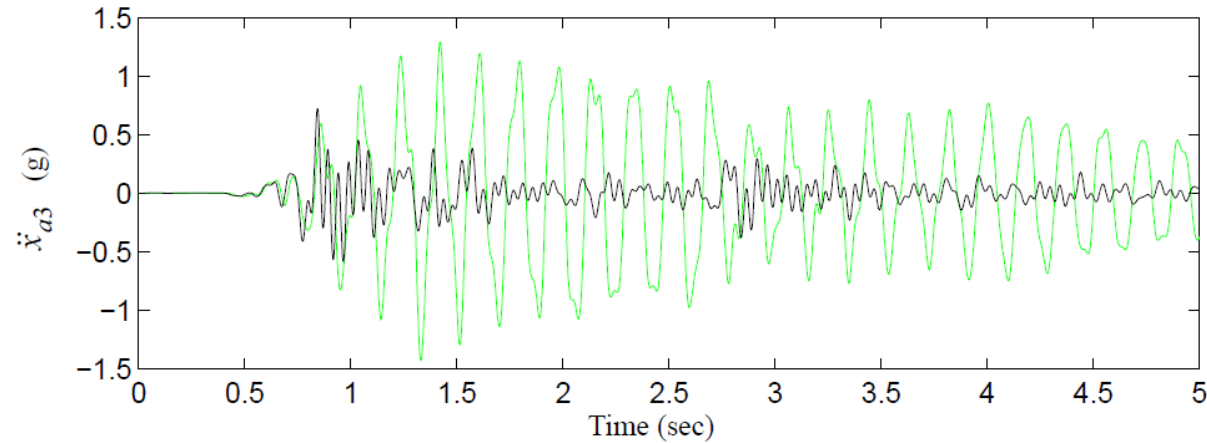
Dynamic analysis of practical building



7. Dynamic package (DynPAK)



7. Dynamic package (DynPan)



Evaluation Criteria

Cairo University

FRBV-Semi Active Evaluation Criteria

J1_eq = 1.0010018244176975
 J2_eq = 0.9936571875212346
 J3_eq = 0.986632446876397
 J4_eq = 1.0069900651425197
 J5_eq = 0.9946355817928664
 J6_eq = 0.9906616492850429
 J7_eq = 3.8402258714533455e-05

Floor	Number of MR Dampers per Floor	Eccentric (2*Ey)
9	No MR Dampers installed	
8	1	No
7	No MR Dampers installed	
6	No MR Dampers installed	
5	1	No
4	No MR Dampers installed	
3	1	No
2	No MR Dampers installed	
1	1	No

7. Dynamic package (DynPak)

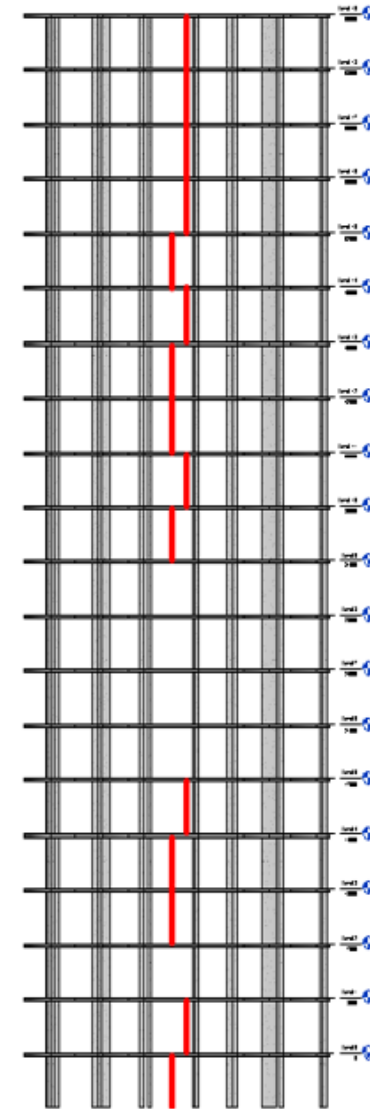
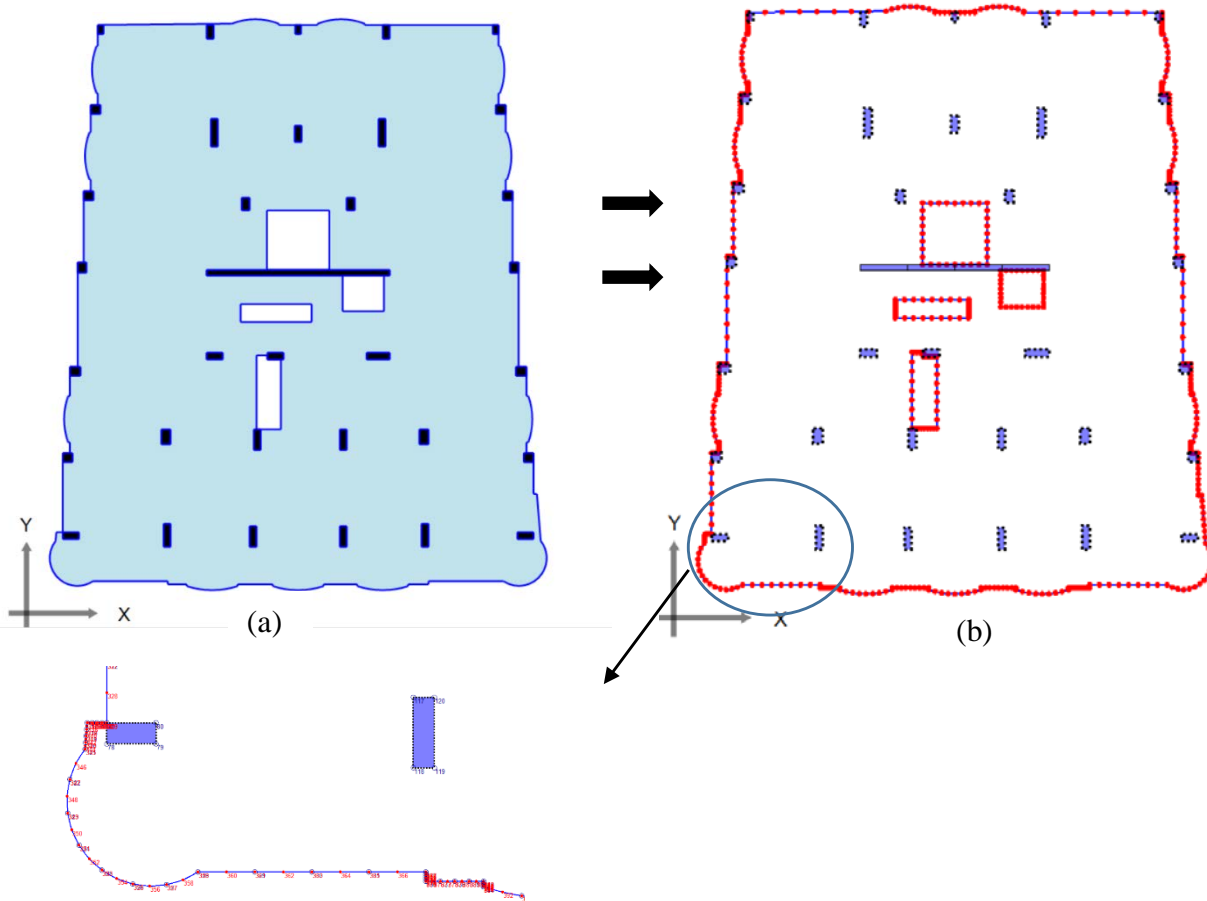


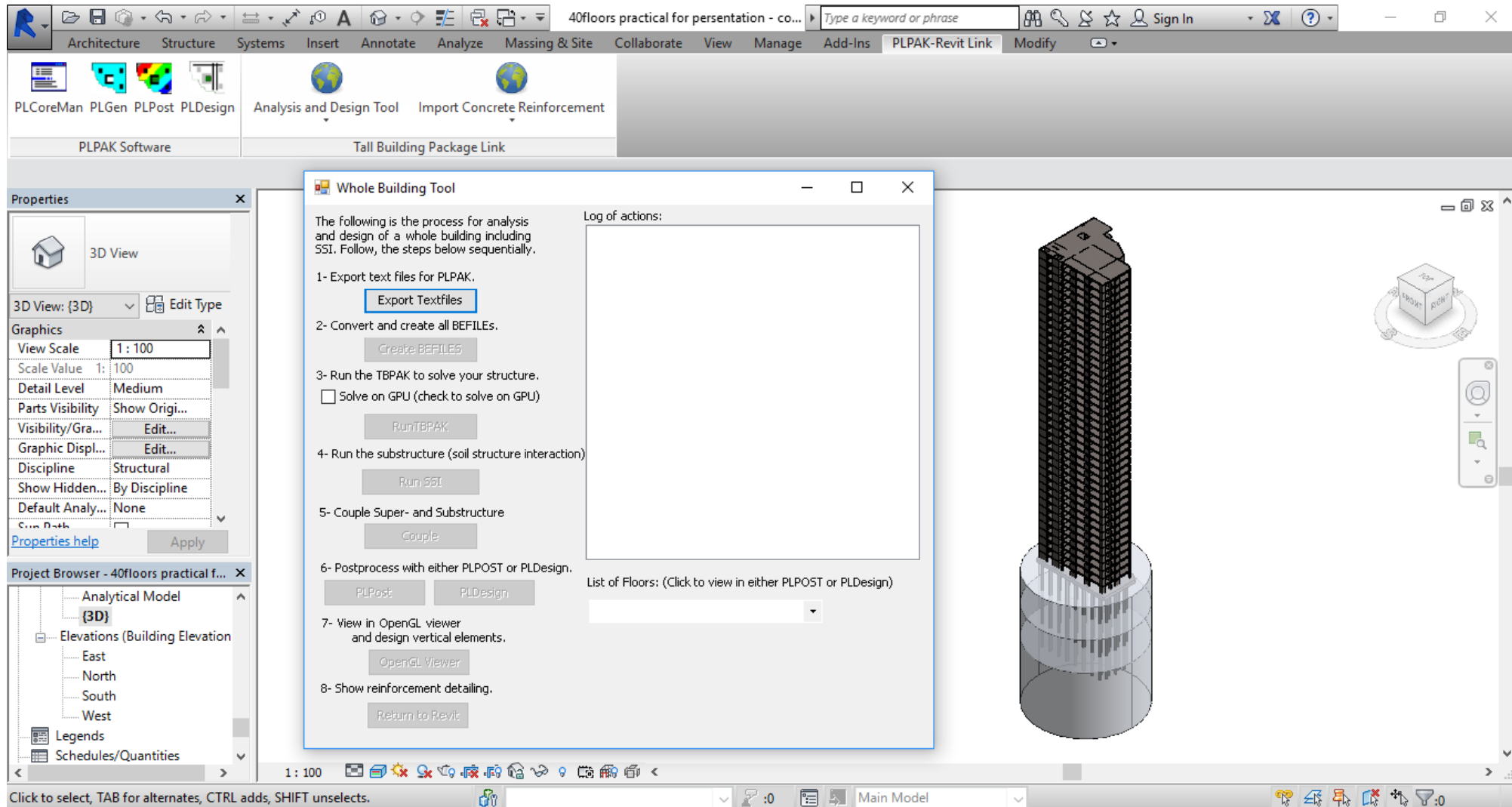
Table 4.8: Genetic Algorithm output for localization of MR dampers for High-Rise Building Model I

Floor	Number of MR Dampers per Floor	Eccentric ($2 \cdot E_y$)
20	7	Yes
19	7	Yes
18	9	Yes
17	9	Yes
16	5	No
15	9	Yes
14	5	No
13	9	No
12	5	Yes
11	7	No
10	<i>No MR Dampers installed</i>	
9	<i>No MR Dampers installed</i>	
8	<i>No MR Dampers installed</i>	
7	<i>No MR Dampers installed</i>	
6	7	Yes
5	9	No
4	7	No
3	<i>No MR Dampers installed</i>	
2	5	Yes
1	9	No

Table of contents

- 1. Introduction**
- 2. Basic package (PLPAK Basic)**
- 3. One floor package (BIM-PLPAK)**
- 4. Foundation Package (FoundPAK)**
- 5. Fixed base Package (FBPAK)**
- 6. Post-tension Package (PTPAK)**
- 7. Dynamic package (DynPAK)**
- 8. Overall building package (OBPAK)**
9. 4D and 5D analysis
10. Conclusions

8. Overall building package (TBPAK)



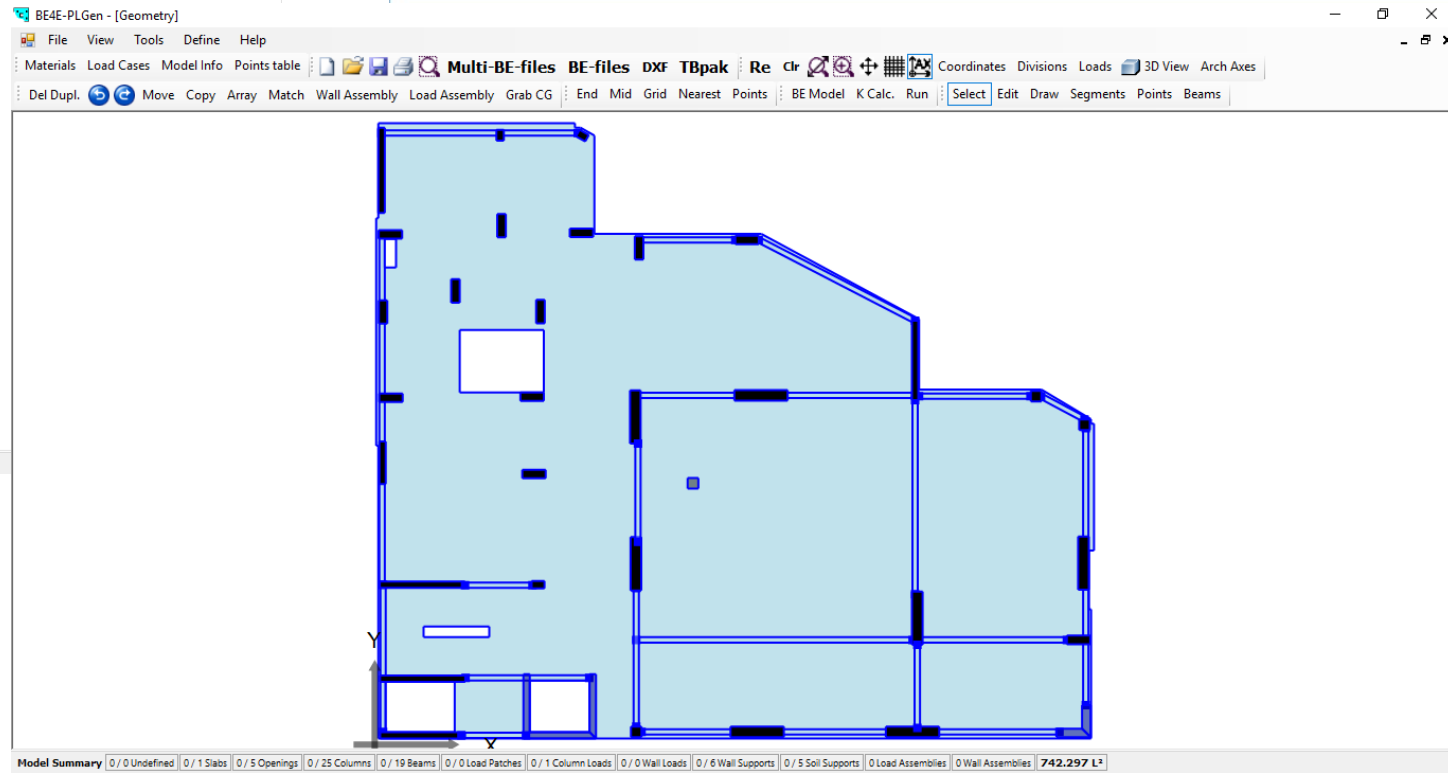
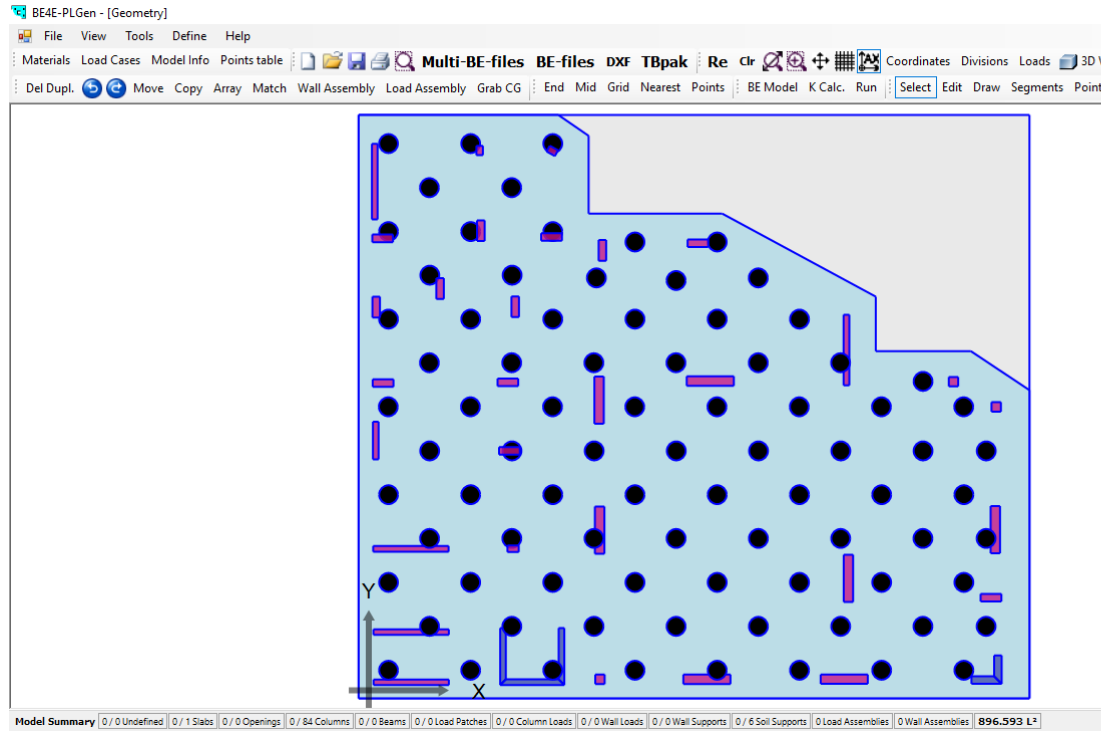
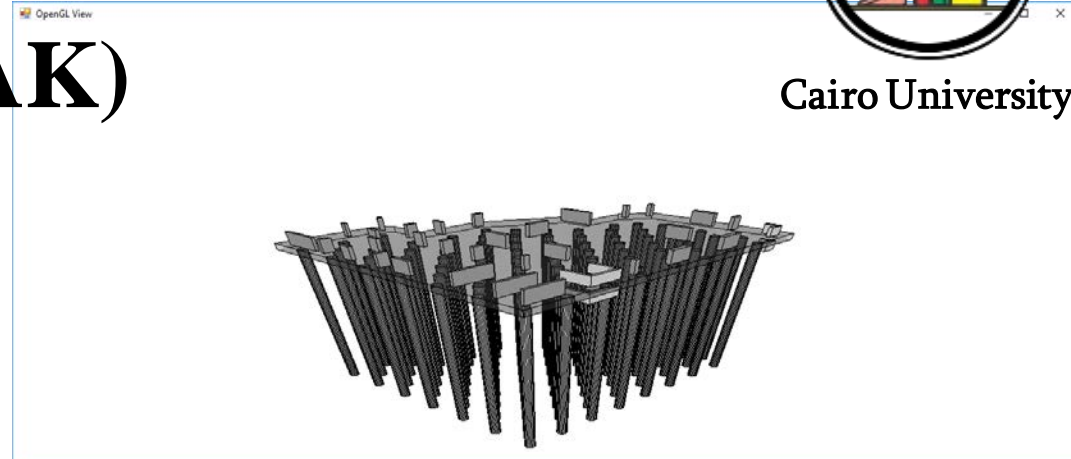
The screenshot displays the PLPAK software interface. The main window shows a 3D model of a tall building. A dialog box titled "Whole Building Tool" is open, providing a step-by-step process for analysis and design. The steps are as follows:

- Export text files for PLPAK. (Button: **Export Textfiles**)
- Convert and create all BEFILES. (Button: **Create BEFILES**)
- Run the TBPAK to solve your structure. (Button: **RunTBPAK**)
 Solve on GPU (check to solve on GPU)
- Run the substructure (soil structure interaction). (Button: **Run SSI**)
- Couple Super- and Substructure. (Button: **Couple**)
- Postprocess with either PLPOST or PLDesign. (Buttons: **PLPost**, **PLDesign**)
- View in OpenGL viewer and design vertical elements. (Button: **OpenGL Viewer**)
- Show reinforcement detailing. (Button: **Return to Revit**)

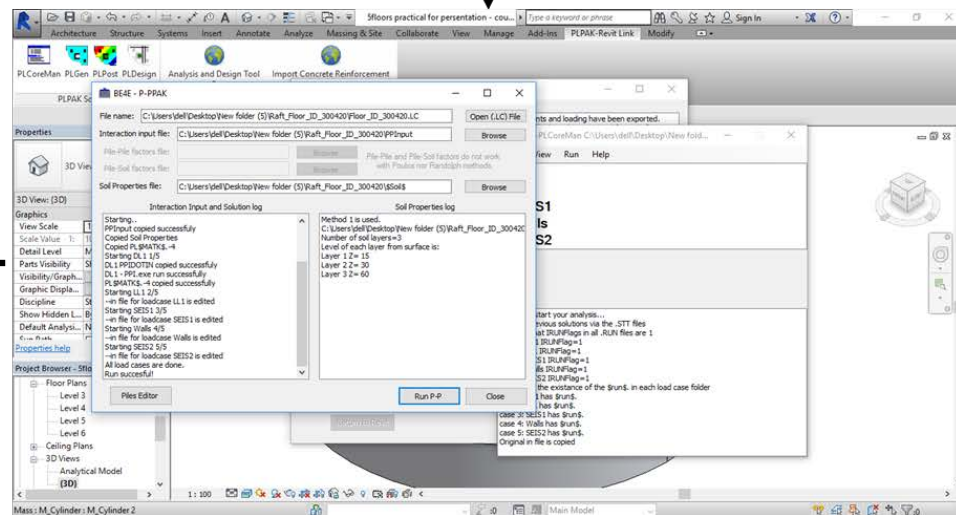
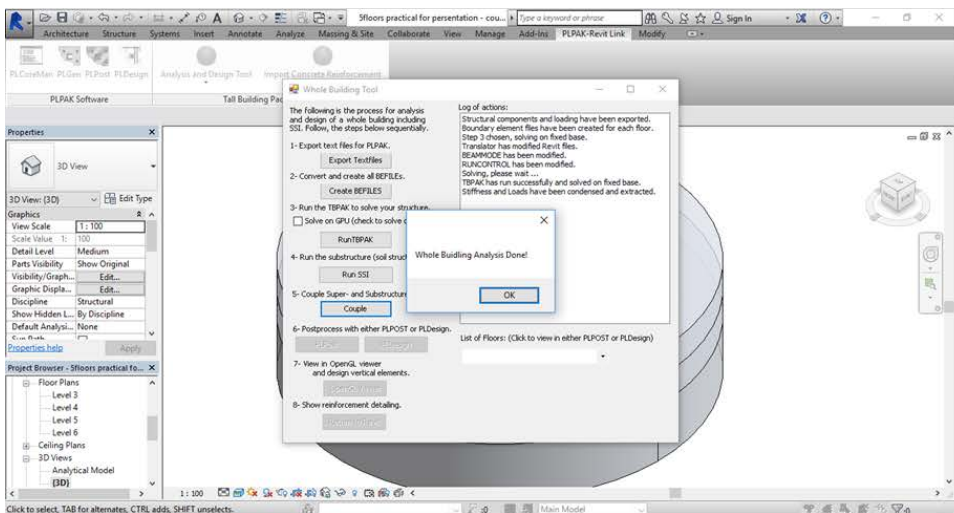
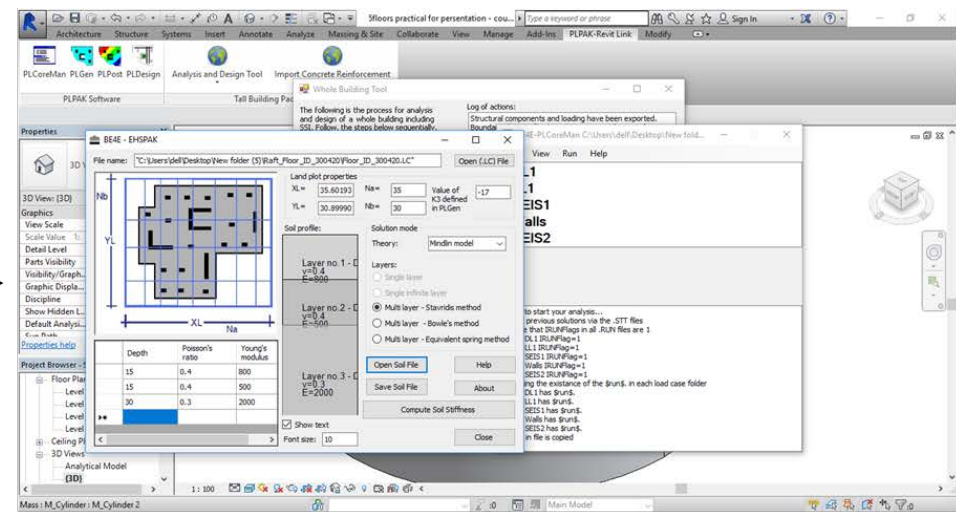
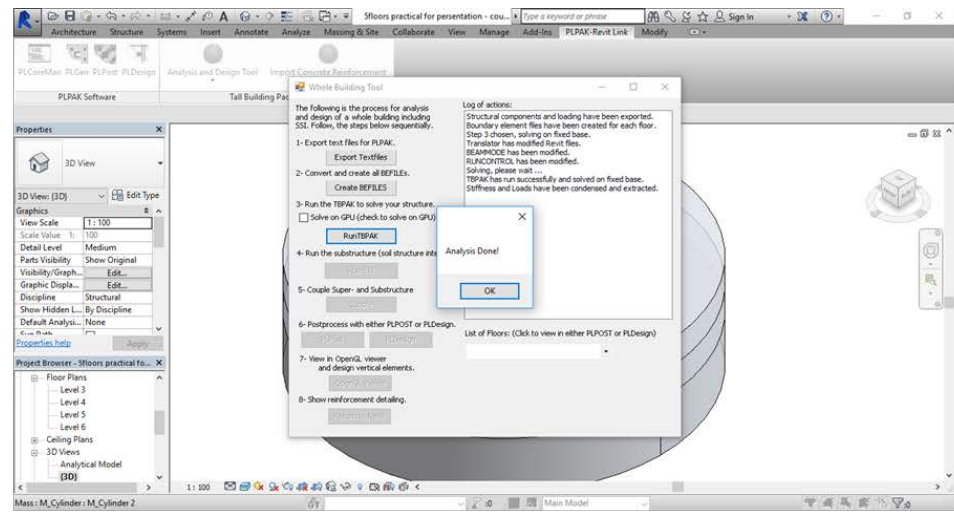
The dialog box also includes a "Log of actions:" text area and a "List of Floors: (Click to view in either PLPOST or PLDesign)" dropdown menu. The software interface includes a ribbon with tabs like Architecture, Structure, Systems, Insert, Annotate, Analyze, Massing & Site, Collaborate, View, Manage, Add-Ins, and PLPAK-Revit Link. A Properties panel on the left shows 3D View settings, and a Project Browser on the bottom left shows the Analytical Model (3D) and Elevations.

8. Overall building package (UBPAK)

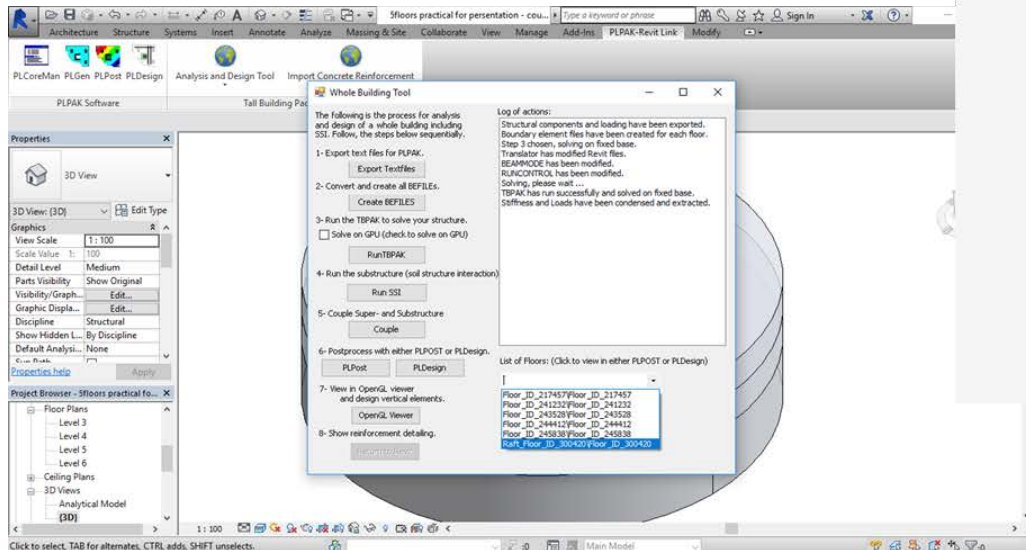
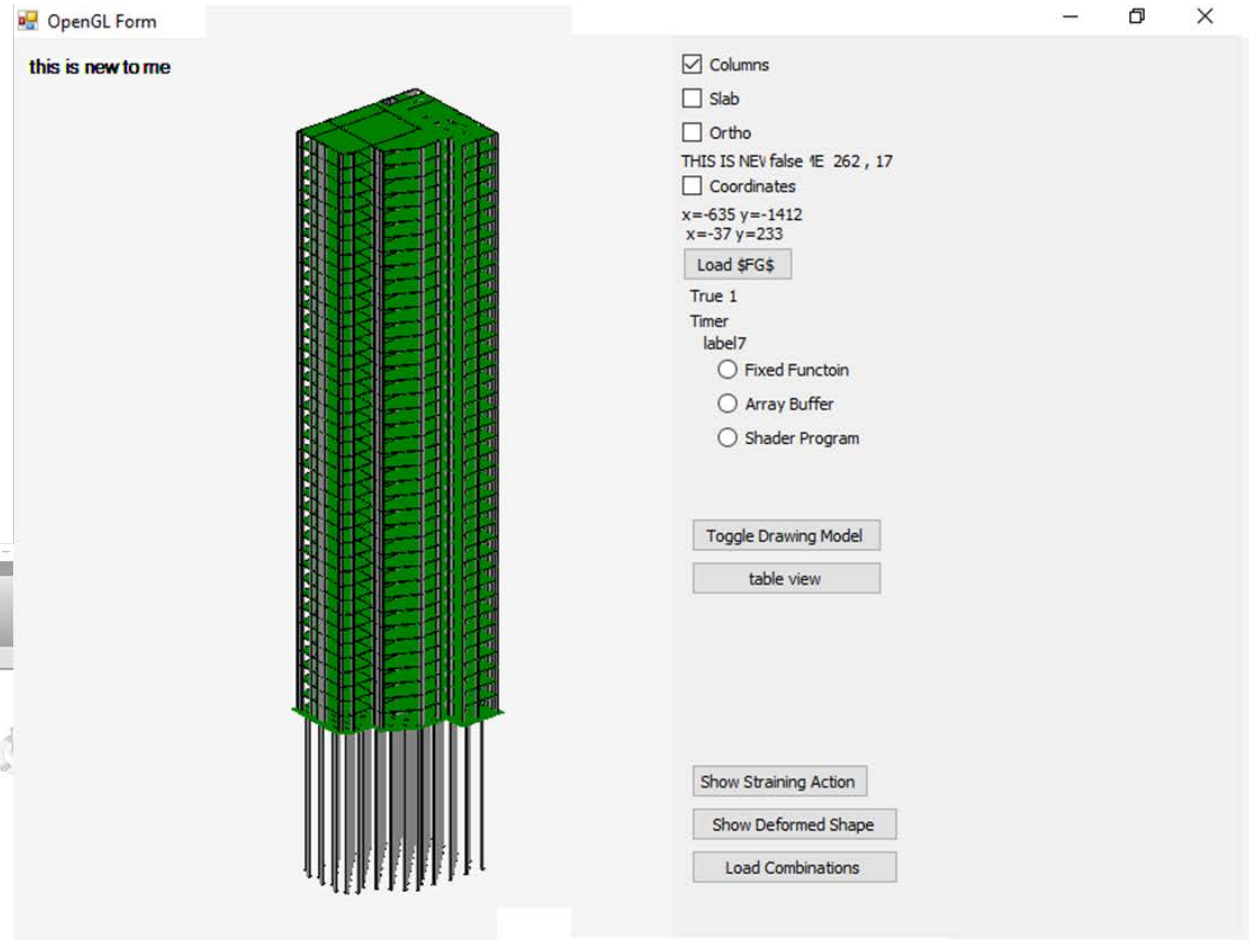
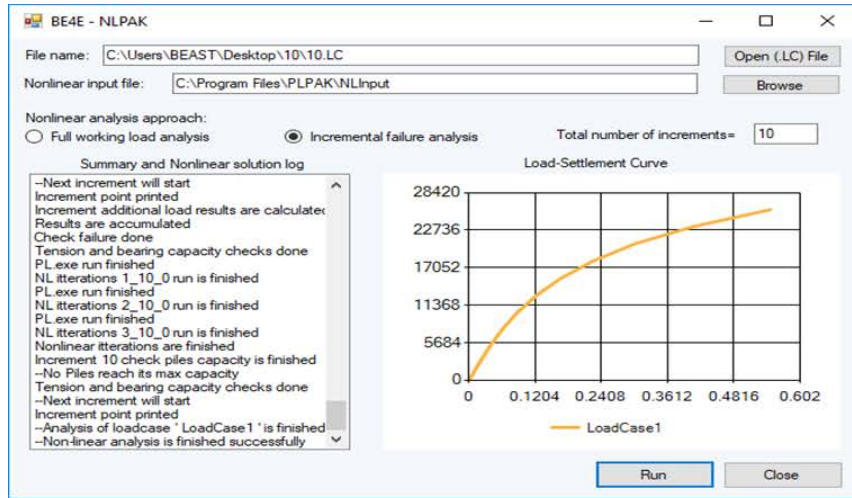
www.b4e.com



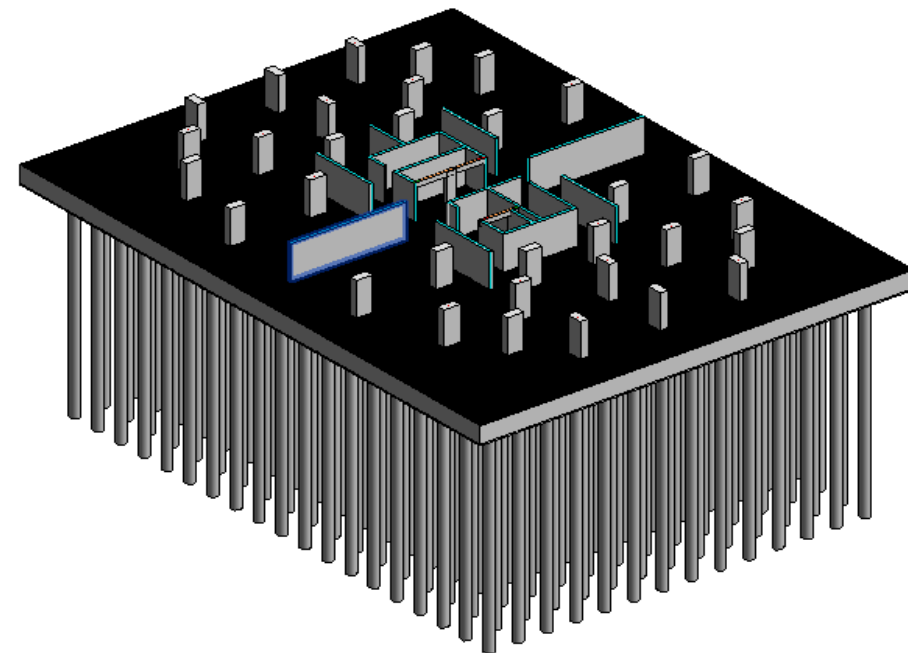
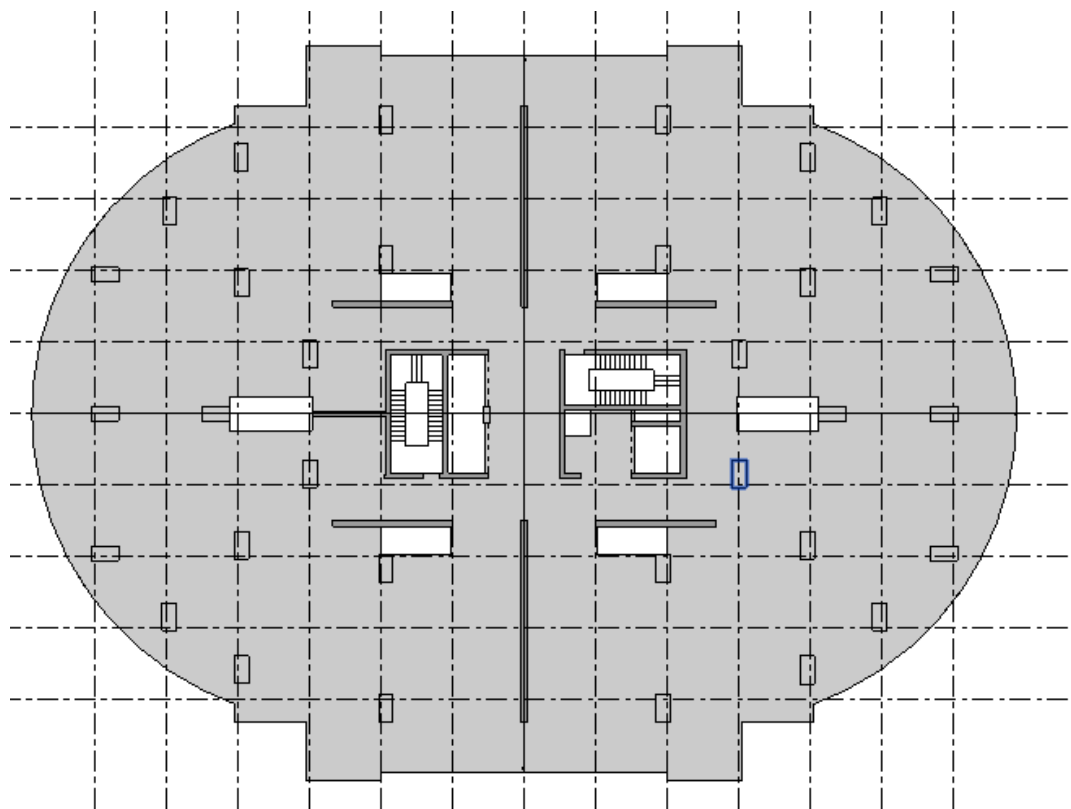
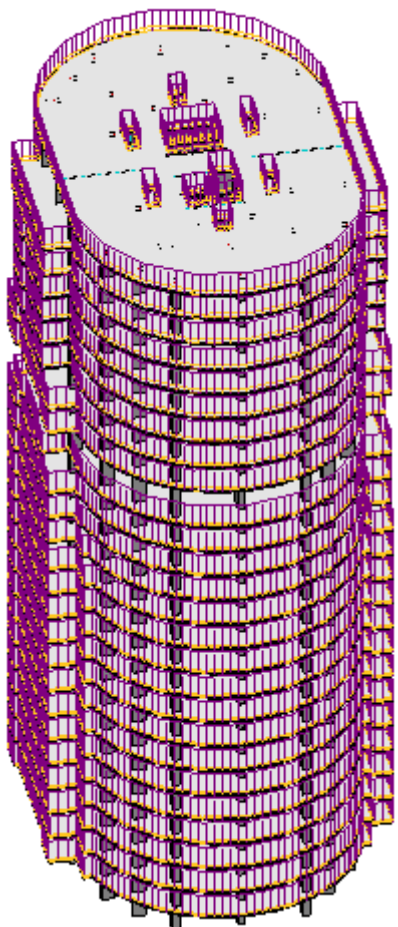
8. Overall building package (UBPAK)



8. Overall building package (UBPAK)

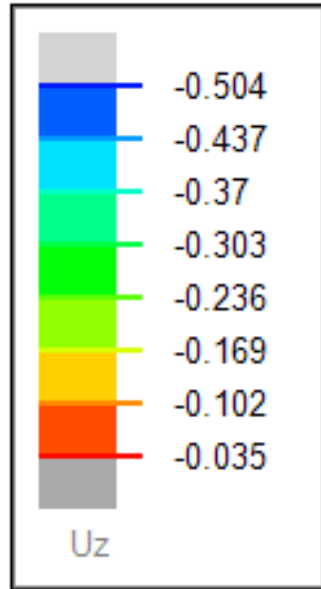
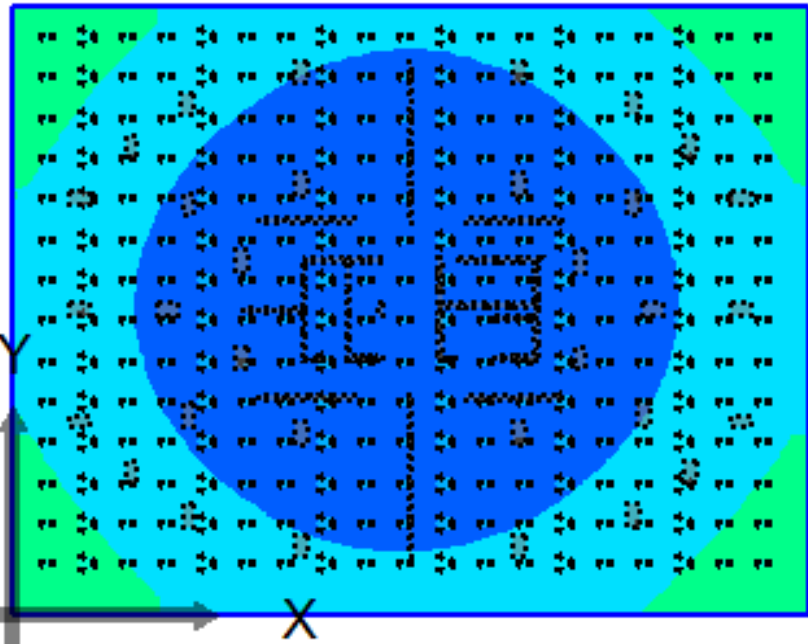


8. Overall building package (UBRAK)

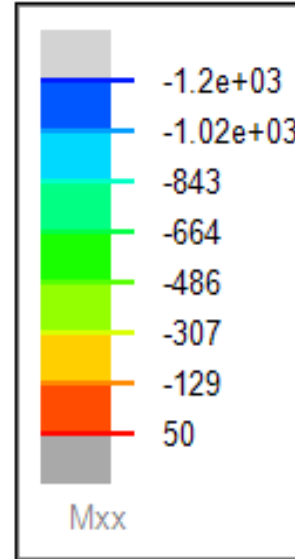
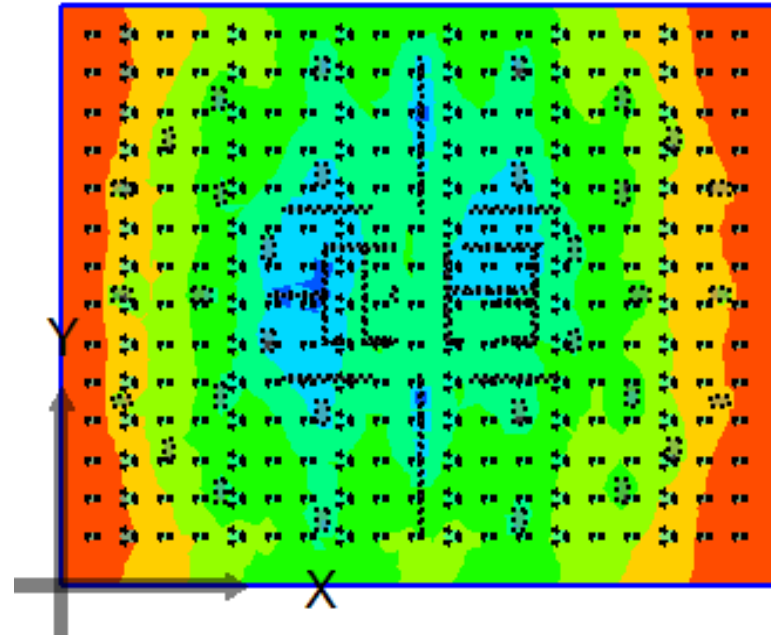


8. Overall building package (UBR-AK)

Cairo University



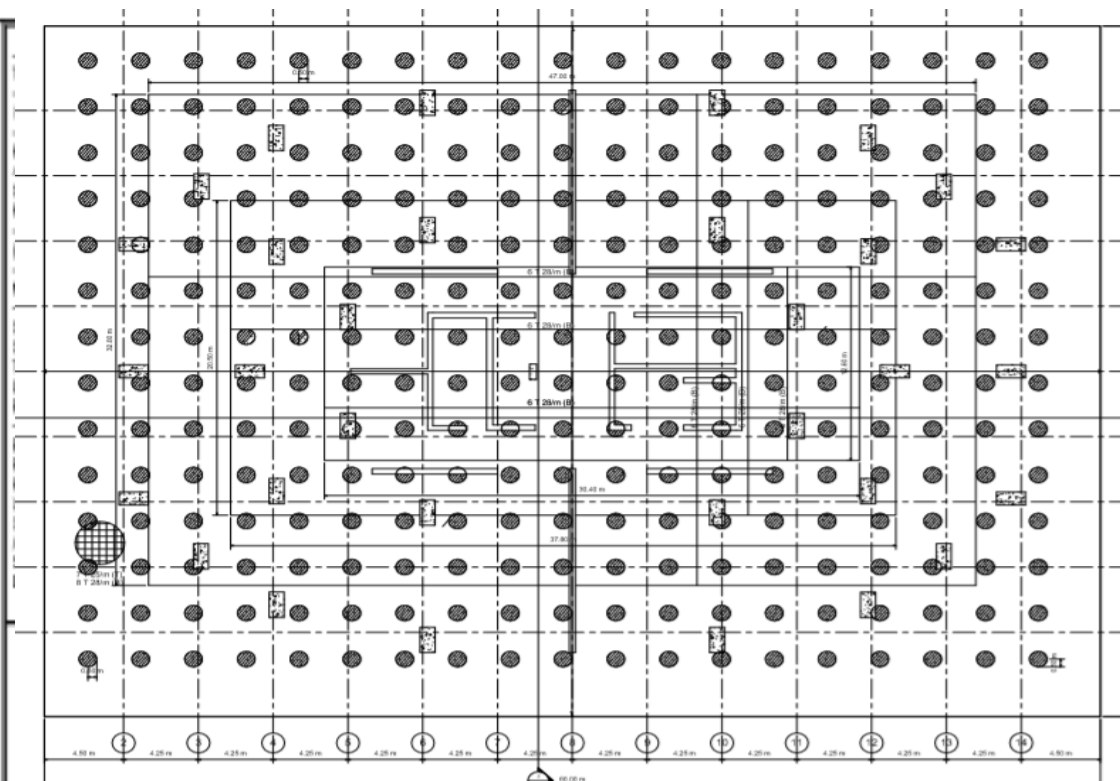
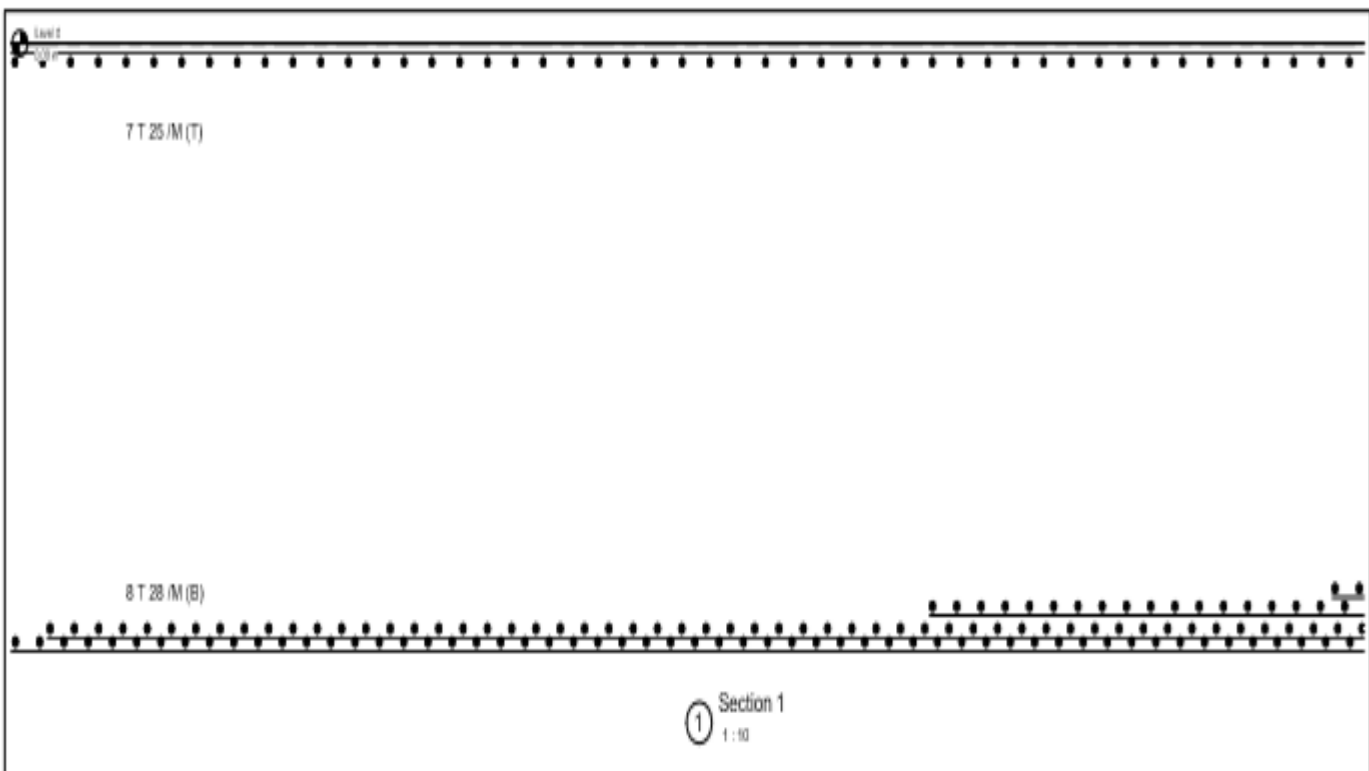
Deflection of raft using p-p interaction.



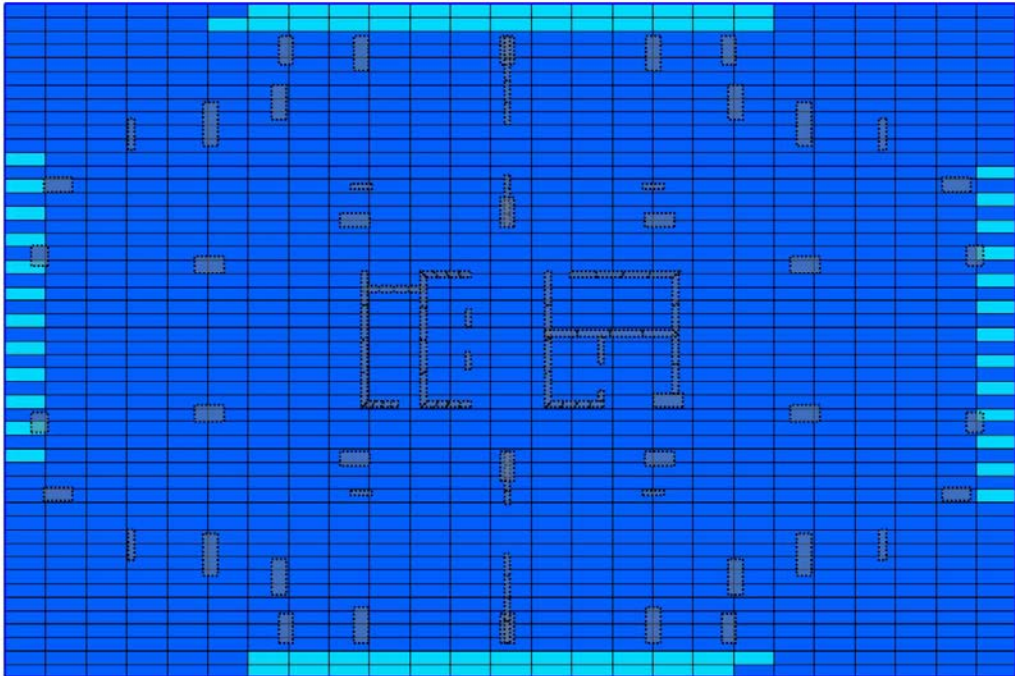
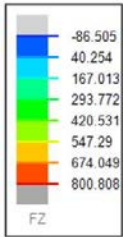
Moment about X-X axis using p-p interaction.

8. Overall building package (UBrAK)

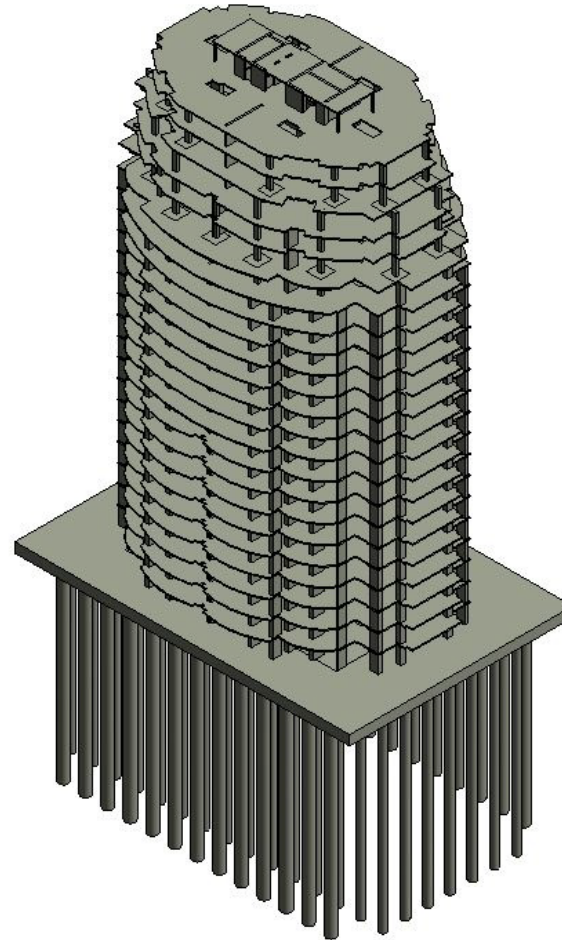
Cairo University



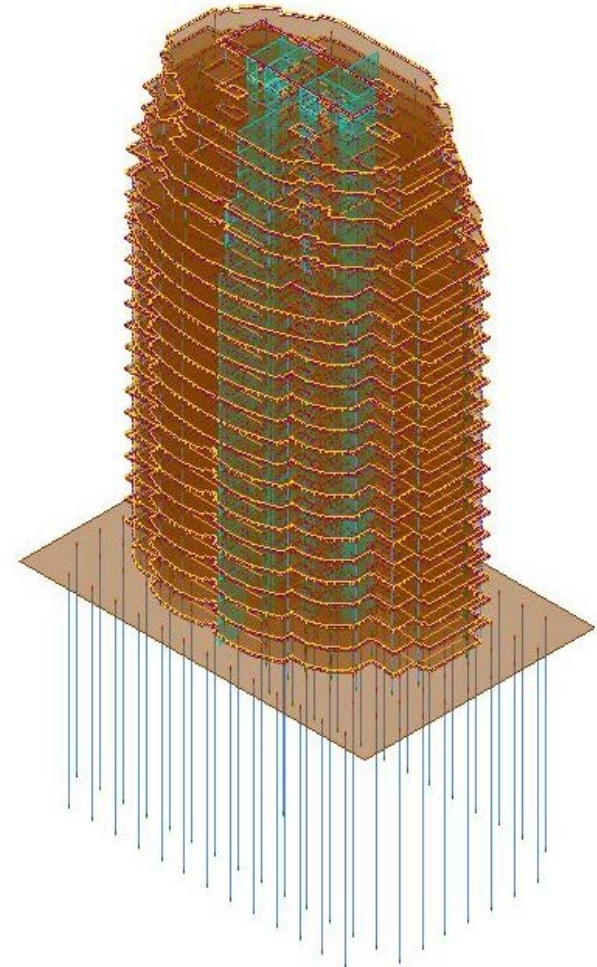
8. Overall building package (UBRAK)



A 38x57 raft fails for the Bearing capacity.

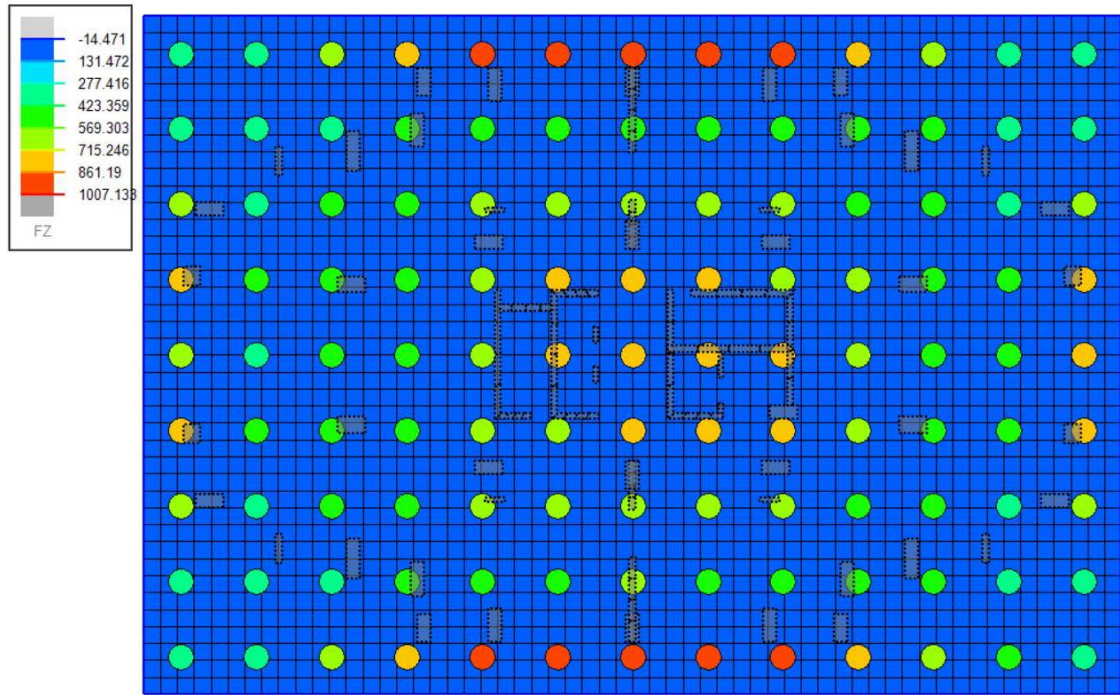


Revit 3D geometric Model

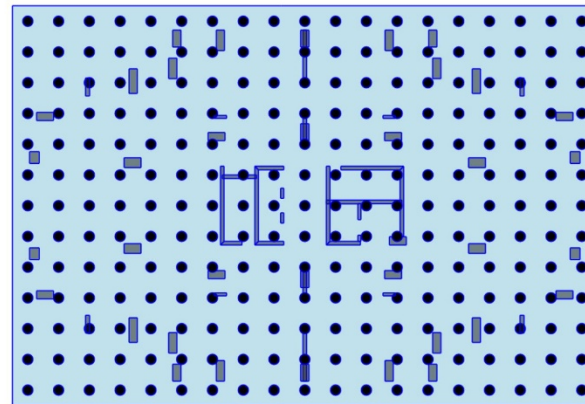


Revit 3D analytical Model

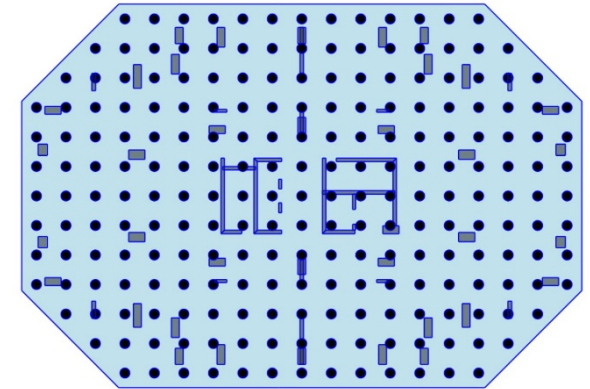
8. Overall building package (UBR-AK)



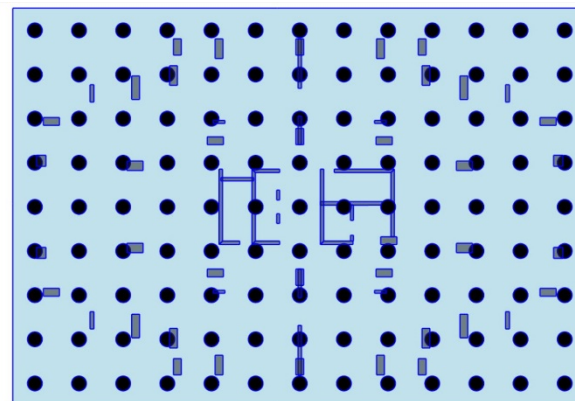
Considering P-P and P-S interactions A piled raft with 117 piles fails against the bearing capacity



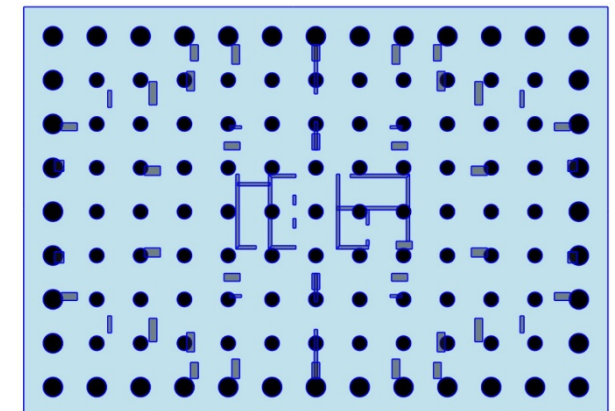
a piled raft with 247 piles.



a raft with 223 piles.

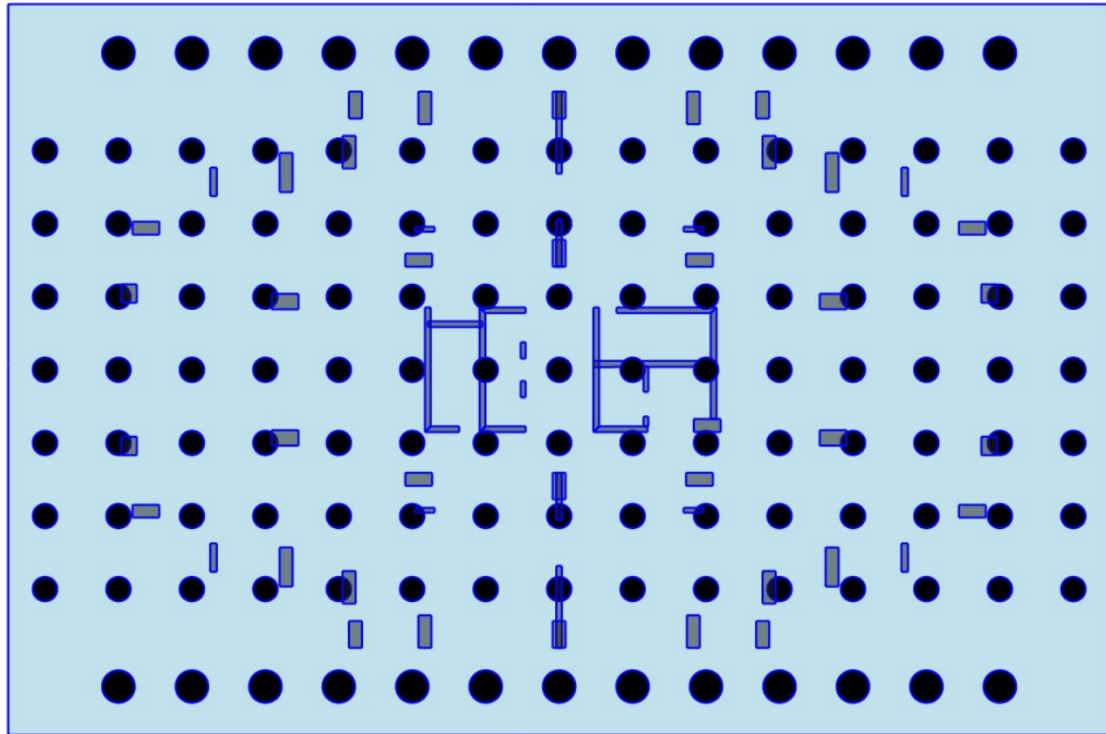


a raft with 117 piles having the same diameter.

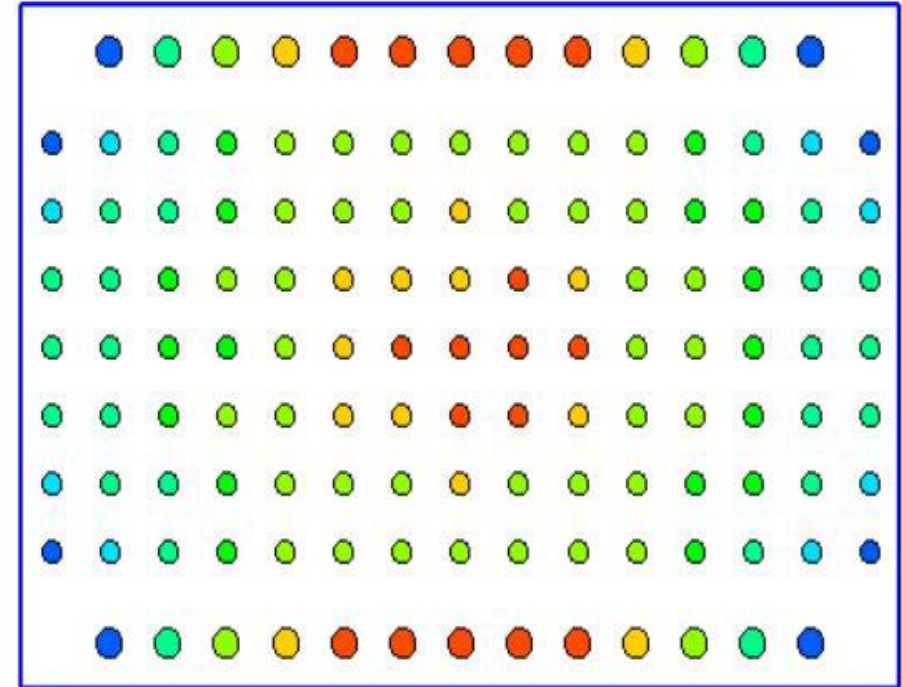
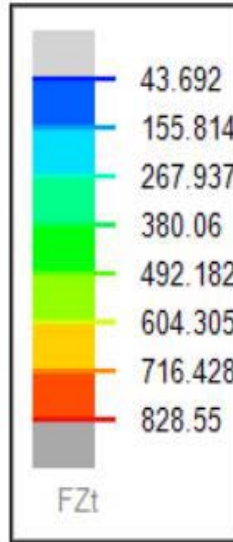


a raft with 117 piles having different diameter

8. Overall building package (www.be4e.com) (K)

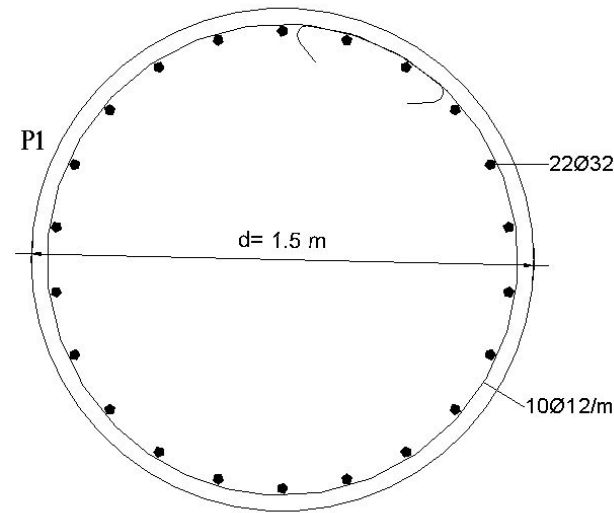
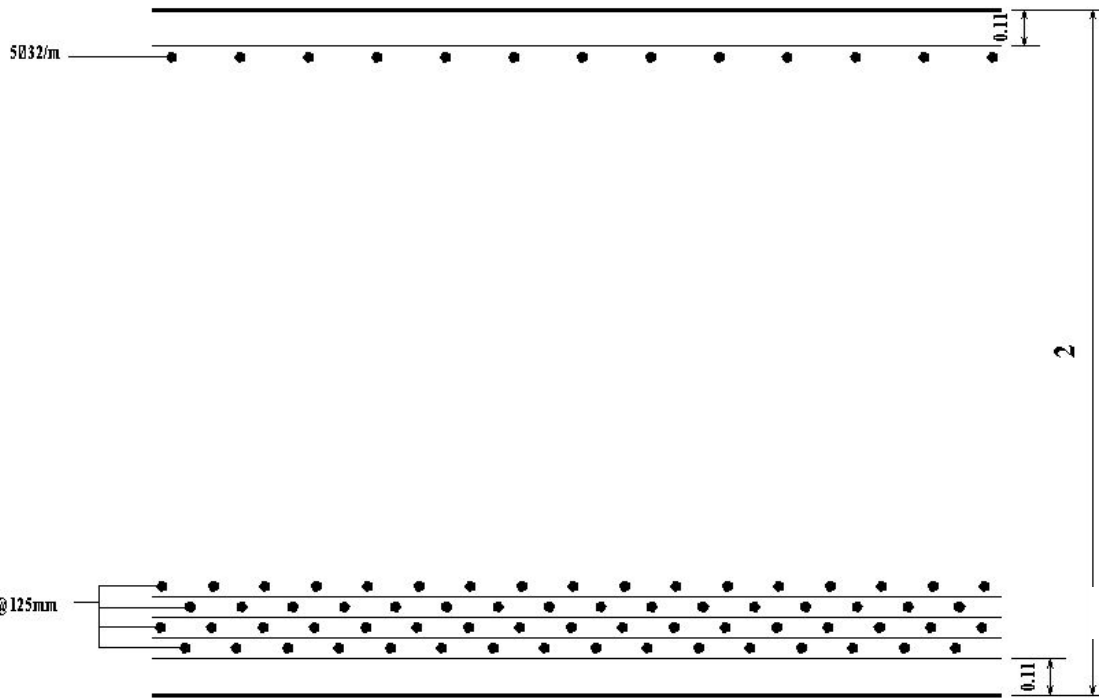


The best configuration for the piled raft with 131 piles having different diameters considering pp only.

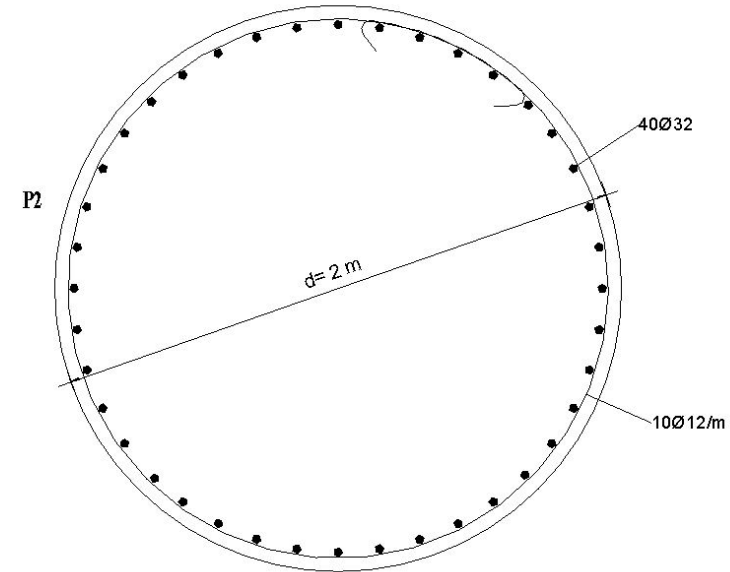


The axial compression straining actions of the piles.

8. Overall building package (UBrAK)



pile#1 x-section with RFT.



pile#2 x-section with RFT.

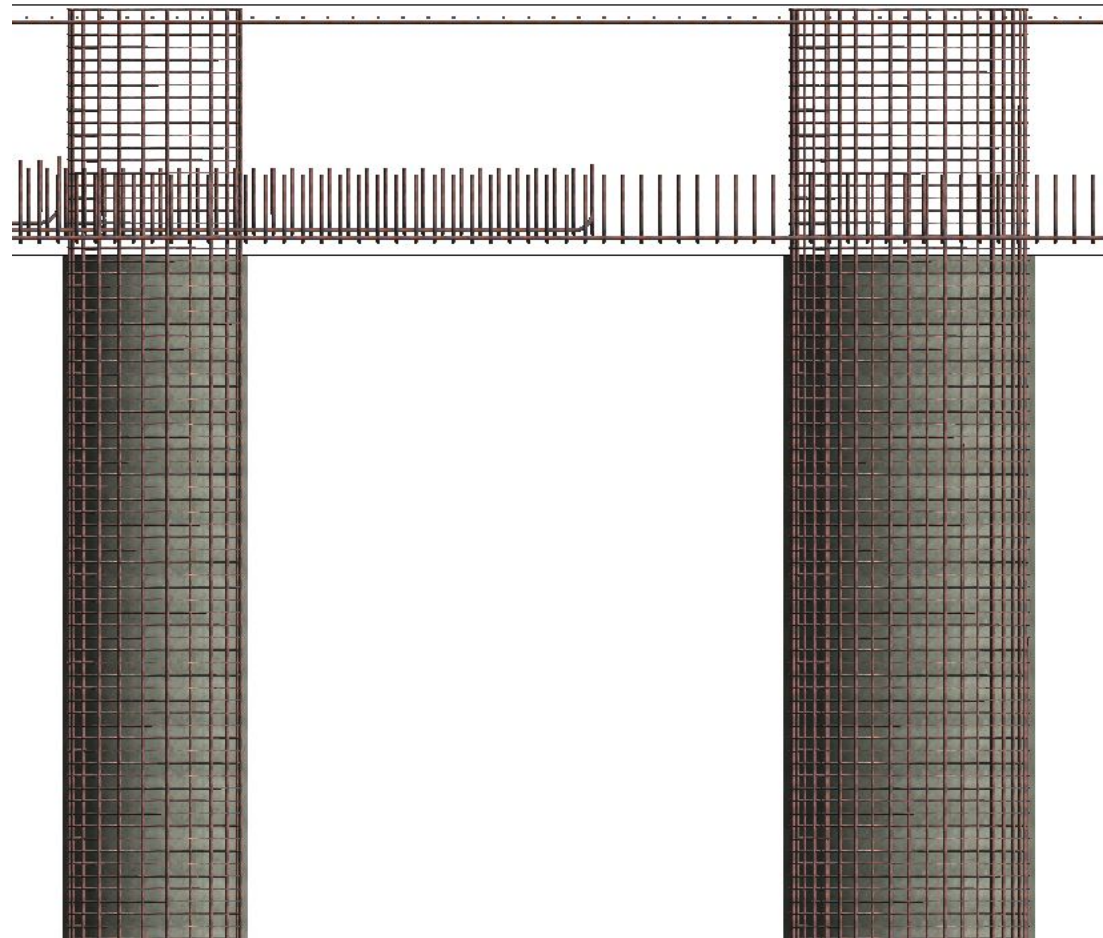
x-section for the piled raft showing the RFT layers.

CUFEBE



Cairo University

8. Overall building package (UBrAK)



Graphical elevation for piles x-sec.

CUFE-BE



www.be4e.com

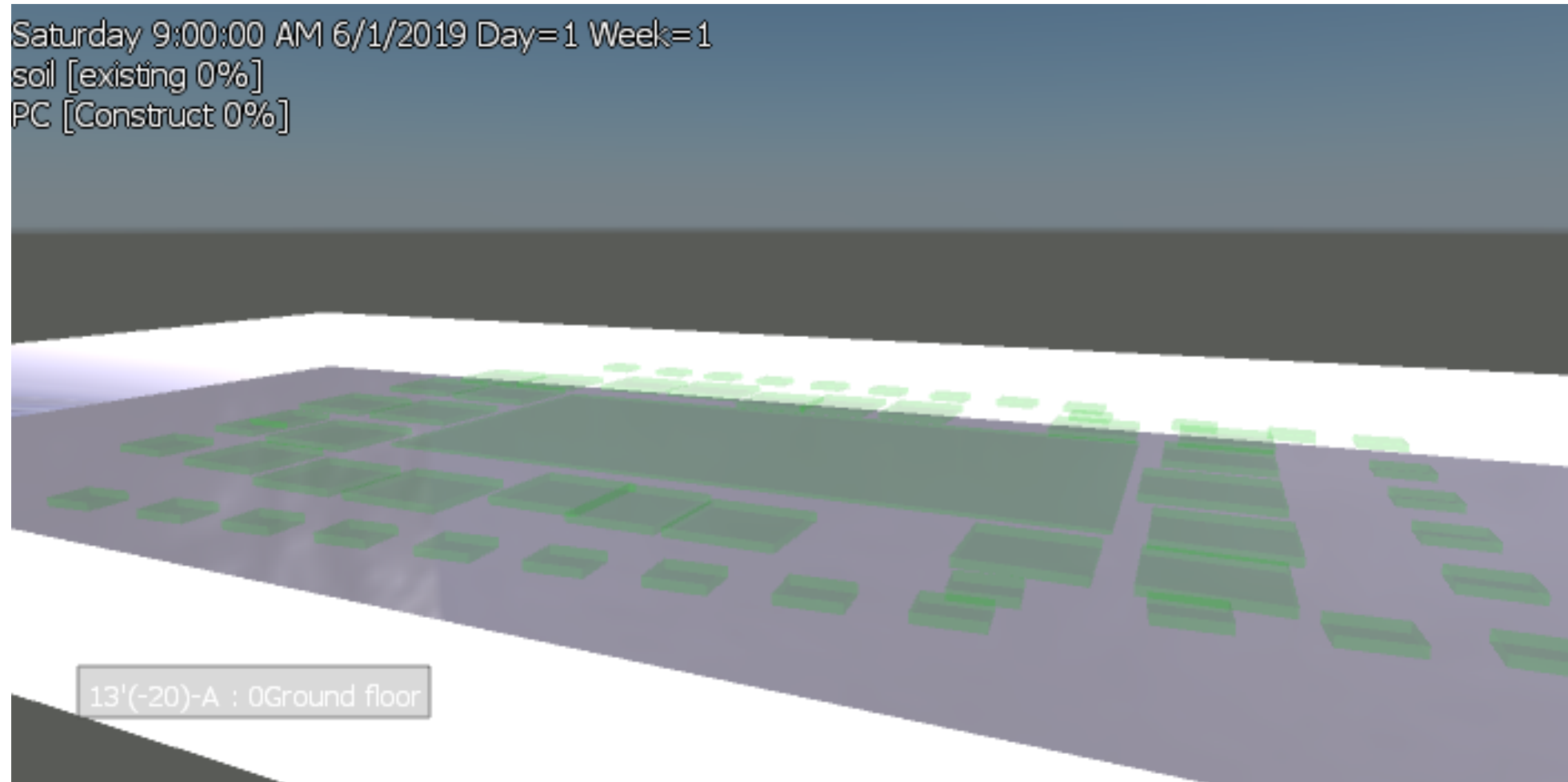


Cairo University

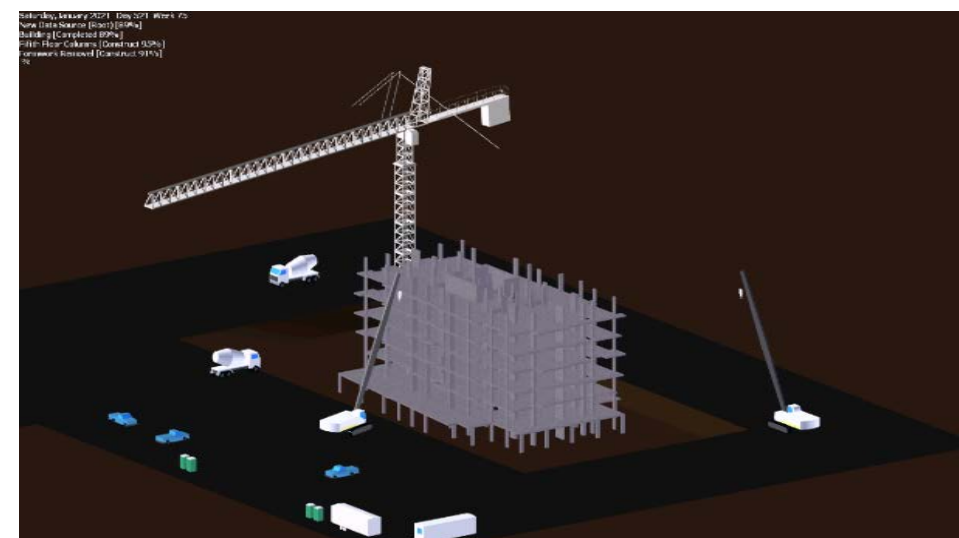
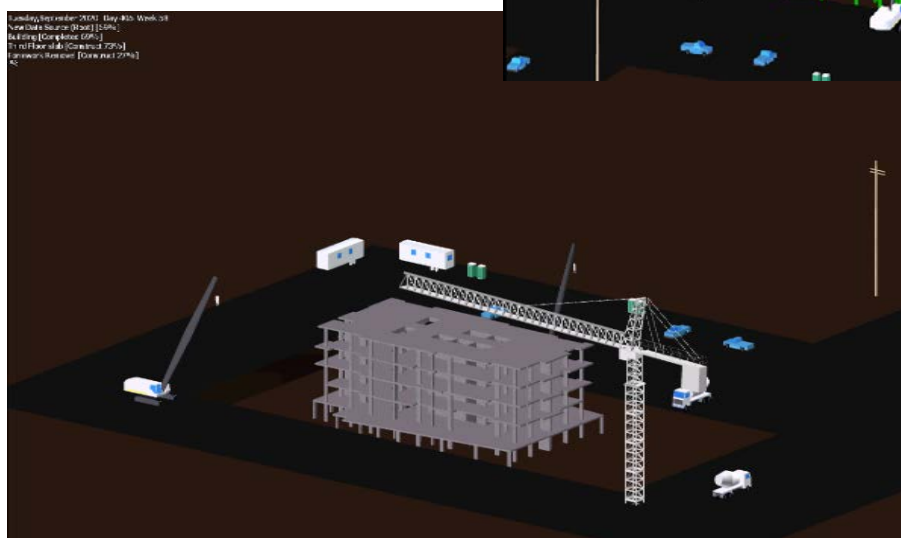
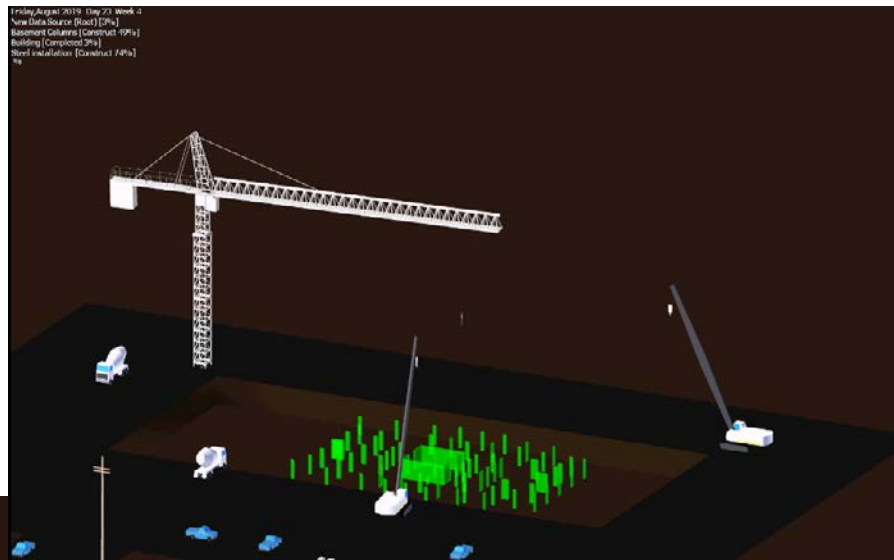
Table of contents

- 1. Introduction**
- 2. Basic package (PLPAK Basic)**
- 3. One floor package (BIM-PLPAK)**
- 4. Foundation Package (FoundPAK)**
- 5. Fixed base Package (FBPAK)**
- 6. Post-tension Package (PTPAK)**
- 7. Dynamic package (DynPAK)**
- 8. Overall building package (OBPAK)**
- 9. 4D and 5D analysis**
- 10. Conclusions**

9. 4D and 5D analysis



9. 4D and 5D analysis

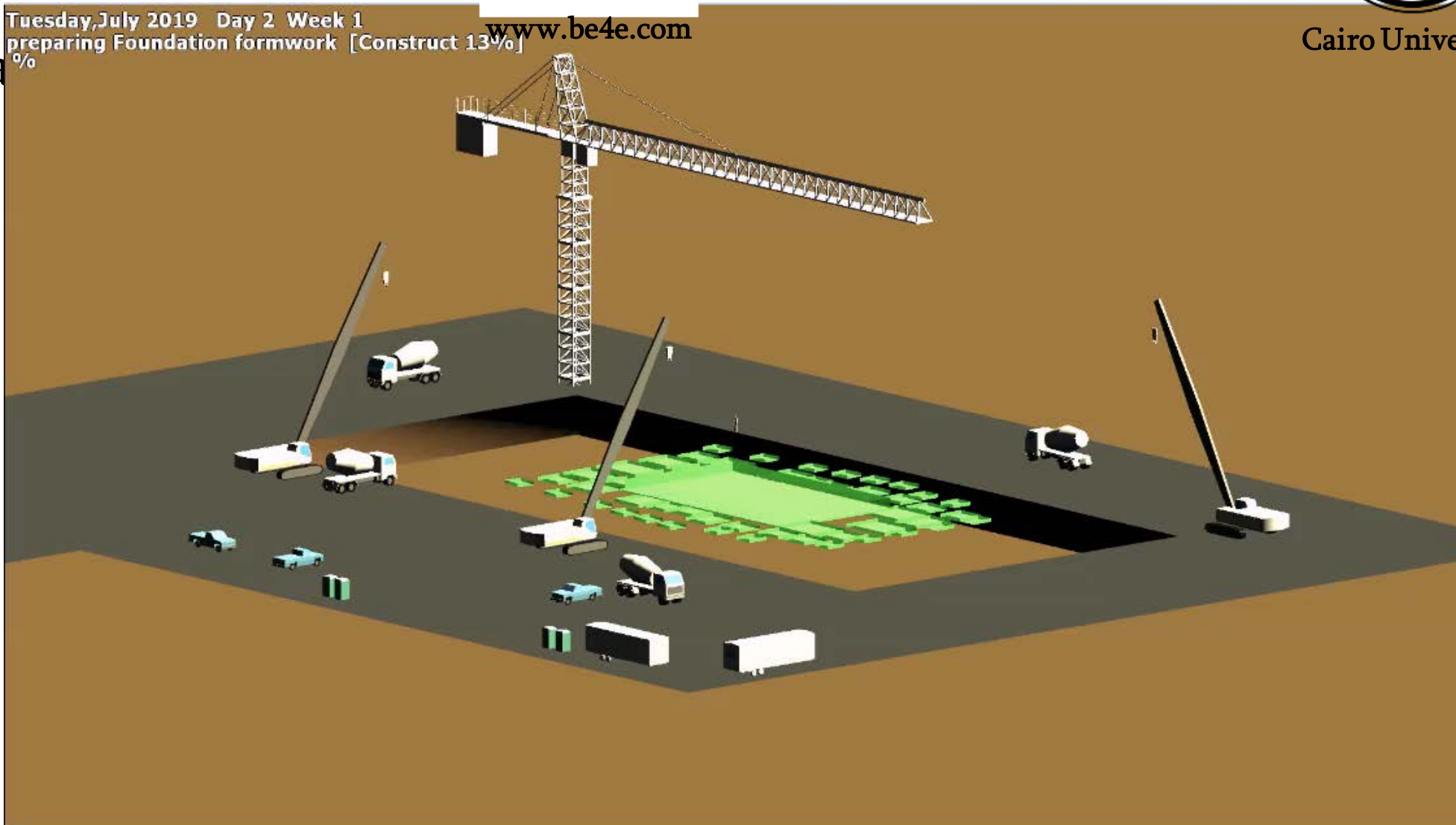


Tuesday, July 2019 Day 2 Week 1
preparing Foundation formwork [Construct 13%]
%

www.be4e.com

Cairo University

9. 4D and 5D a



CUFE-BE



www.be4e.com



Cairo University

Table of contents

- 1. Introduction**
- 2. Basic package (PLPAK Basic)**
- 3. One floor package (BIM-PLPAK)**
- 4. Foundation Package (FoundPAK)**
- 5. Fixed base Package (FBPAK)**
- 6. Post-tension Package (PTPAK)**
- 7. Dynamic package (DynPAK)**
- 8. Overall building package (OBPAK)**
- 9. 4D and 5D analysis**
- 10. Conclusions**

Thanks Any questions ?