CUFE-BE

www.be4e.com

Lecture (7)

PLPAK – Basic Package

2019

CUFE-BE

www.be4e.com

Table of contents

Intended learning outcomes

By the end of this lecture the attendee will learn how to:

1. **Model practical slabs using PLGen.**
2. **Model raft over Winkler area springs using PLGen.**
3. **Check the numerical model using PLView.**
4. **Run the model using PLCoreMan.**
5. **Show results – Strip / contours / reactions / assemblies results using PLPost.**
6. **Show beam results using PLPost.**
7. **Learn PLPAK modelling capabilities.**

CUFE-BE

www.be4e.com

Table of contents

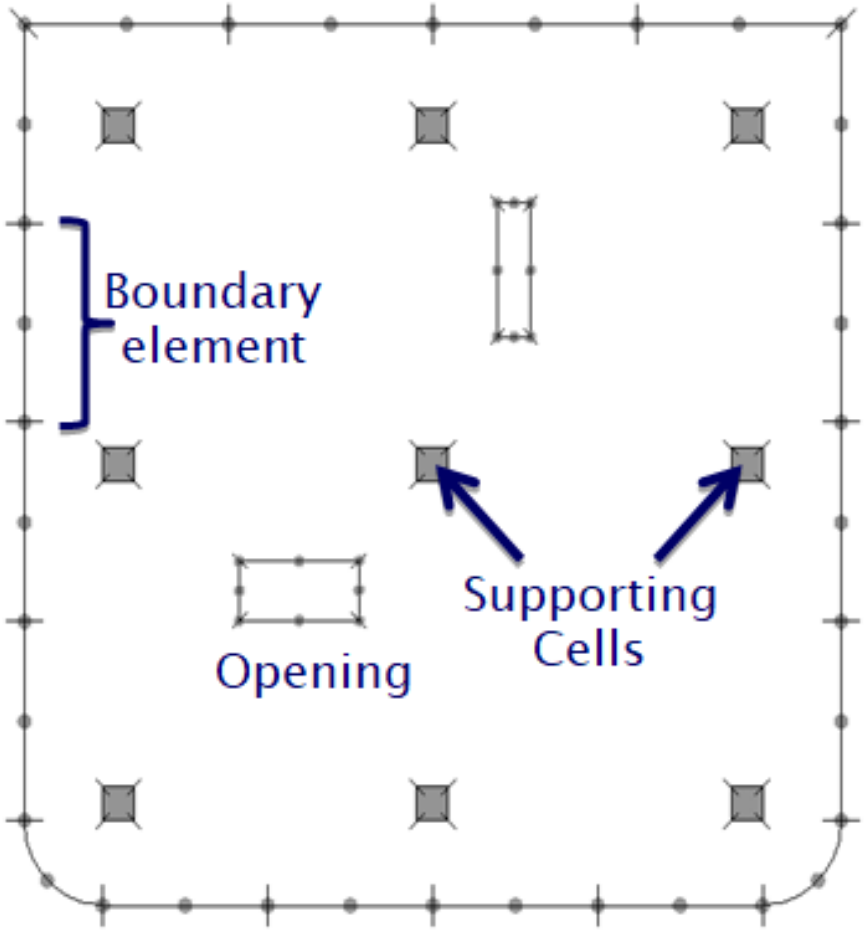
What is the PLPAK?

- 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.

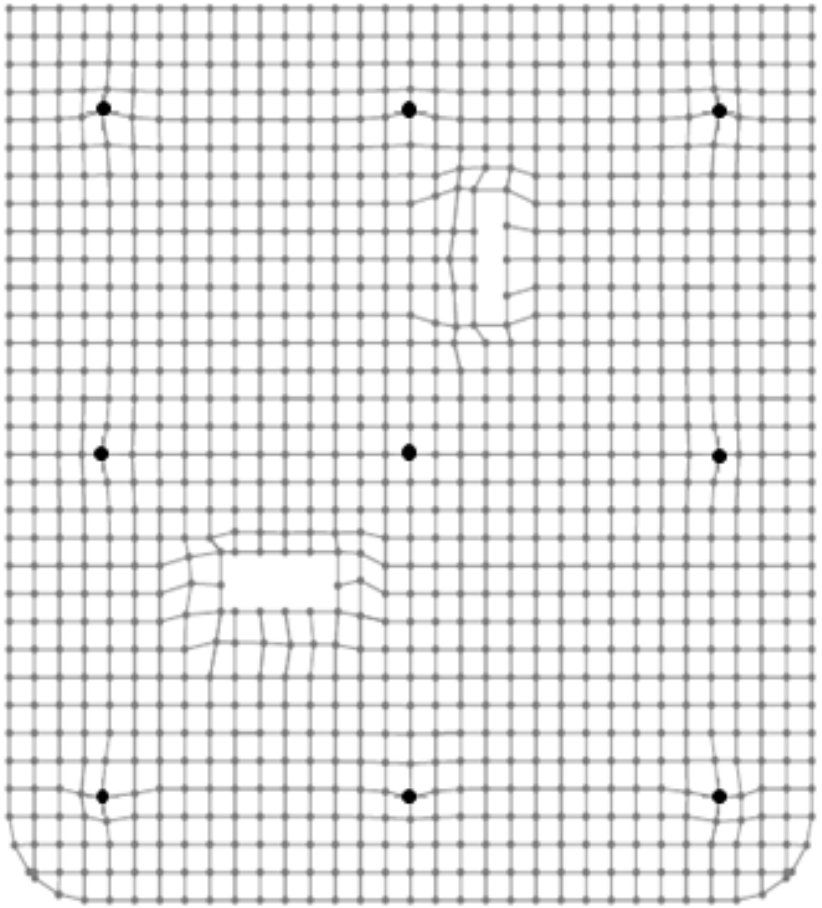
CUFE-BE

www.be4e.com

BEM vs FEM



BEM model



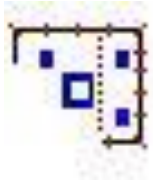
FEM model

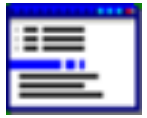
CUFE-BE

www.be4e.com

PLPAK Basic – package components:

1- Generator (PLGen). 

2- Boundary element numerical model viewer (PLView). 

3- Manager (PLCoreMan). 

4- Post-processing (PLPost). 

CUFE-BE

www.be4e.com

Table of contents

CUFE-BE

www.be4e.com

Table of contents

The PLGen is responsible for generating the model showing every structu

Generating the model is in main three steps:

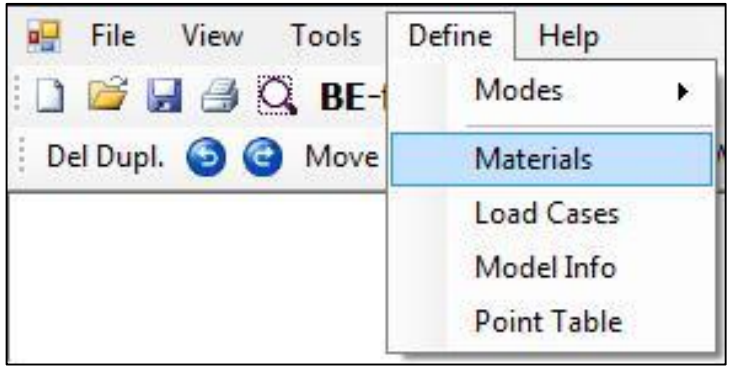
1. **Edit model information**

2. **Build model and define its elements**

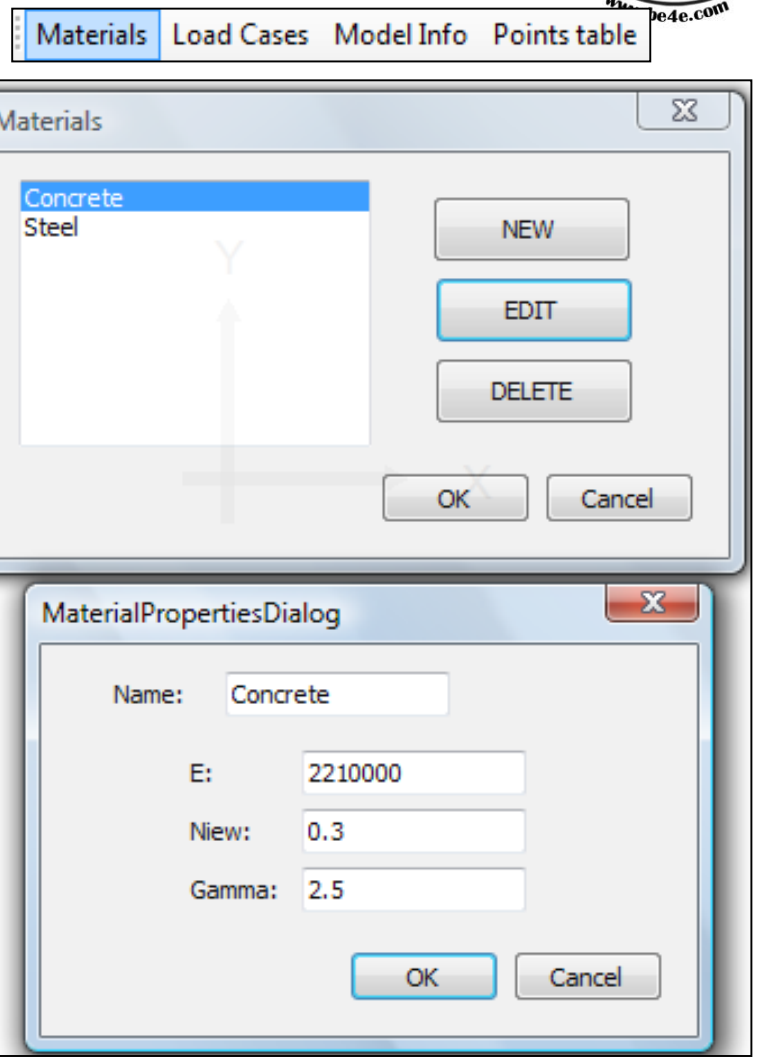
3. **Edit the boundary element divisions**

1. Edit model information

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



OR

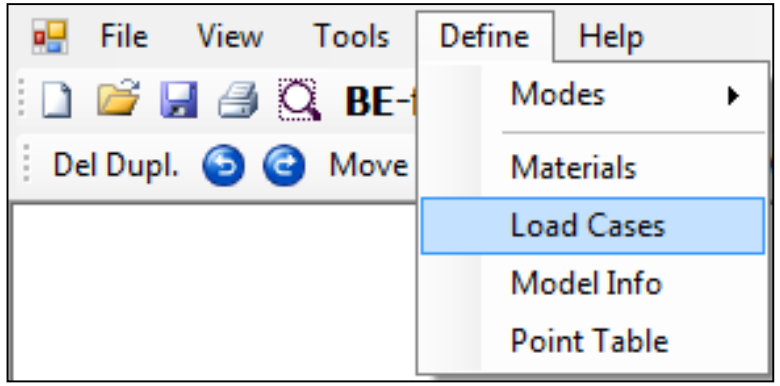


The user can define the properties of the construction material such as Young's modulus, Poisson's ratio and Gamma of the materials.

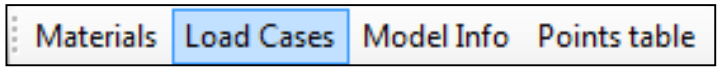
It has to noted that PLPAK is a dimensionless software, that is mean that the user should input the information in a compatible units.

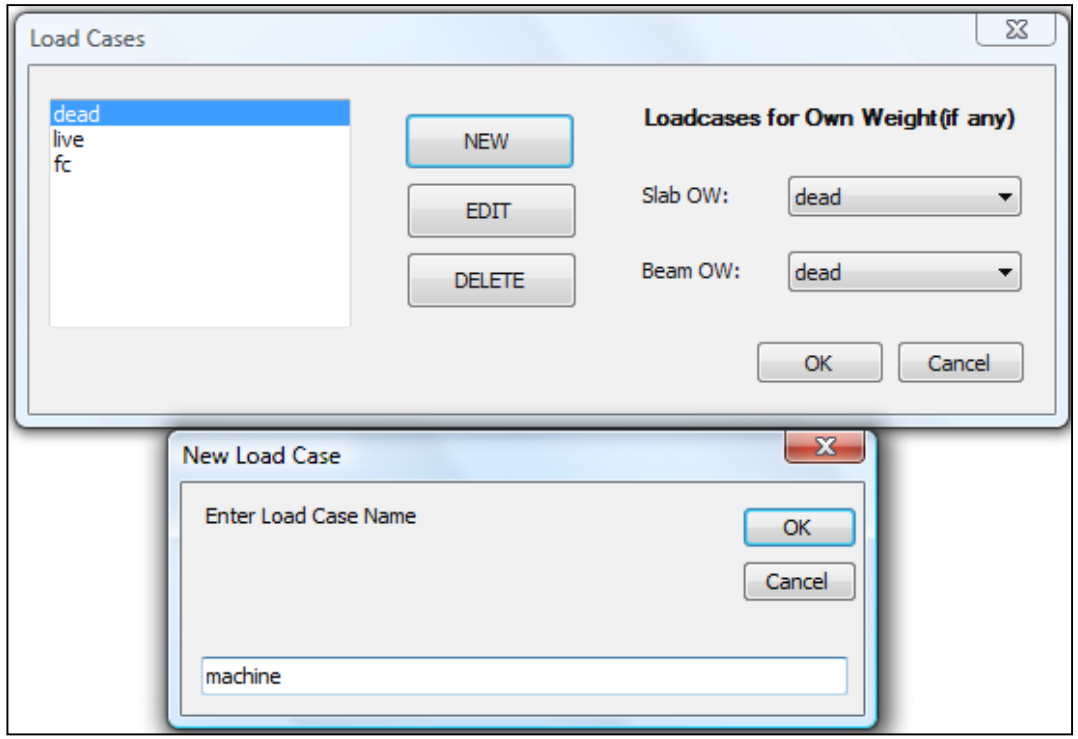
1. Edit model information

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



OR

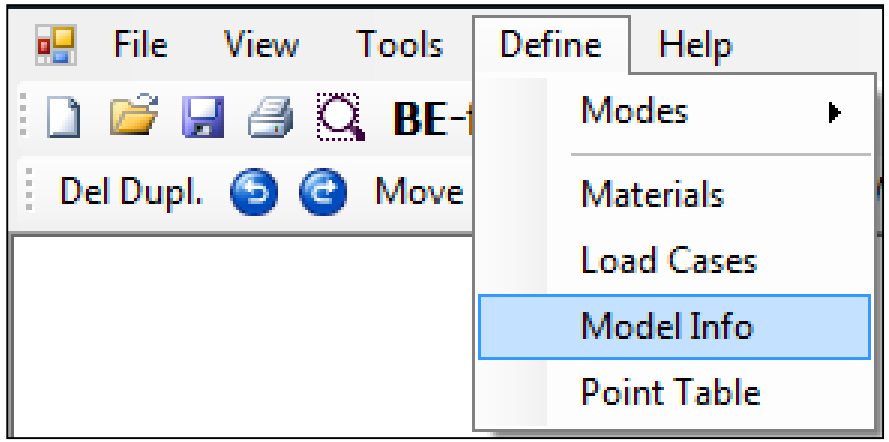




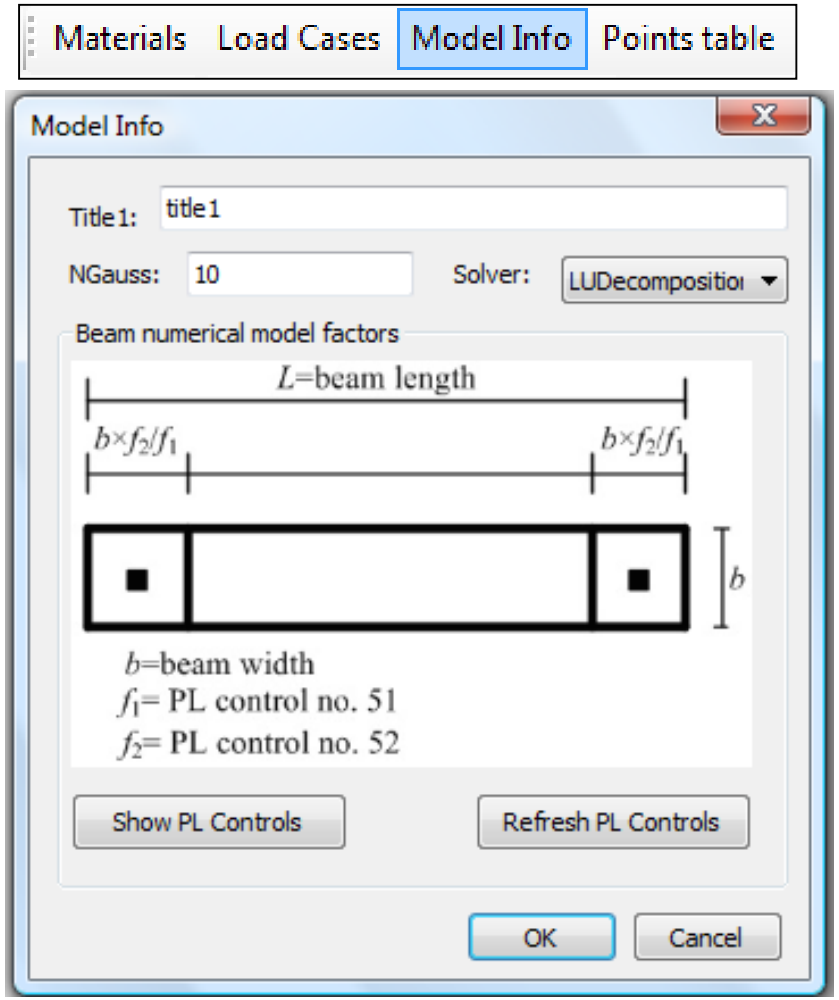
The user can define different load cases and which one can include the slab and beam own weight.

1. Edit model information

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



OR

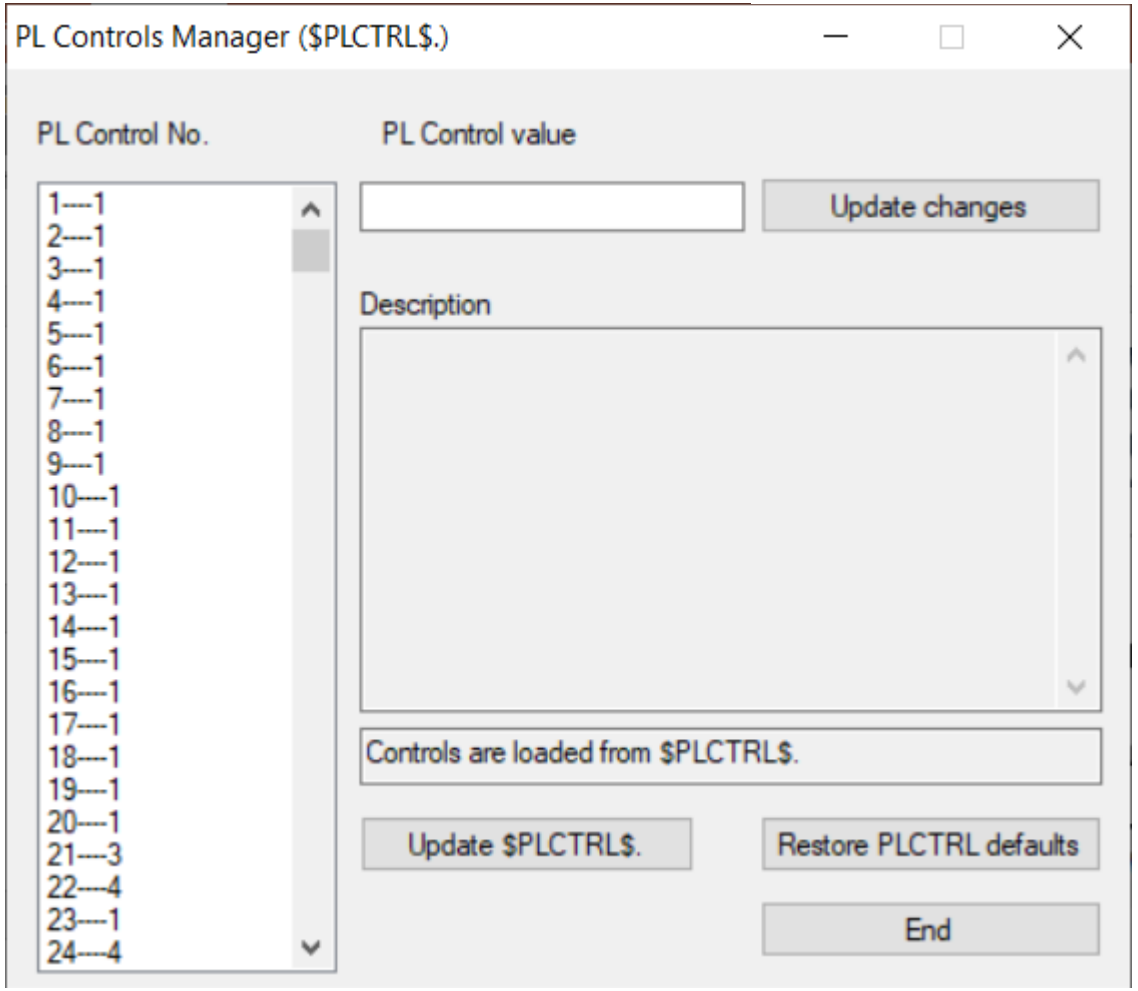


- The user can change number of Gauss points to decrease time consumption in modeling but the number should be <u>even number</u>.
- The user can change the type of solver either LU Decomposition, Gauss Elimination, or GPU solver.
- Also the user can change the PL Controls.

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.

CUFE-BE

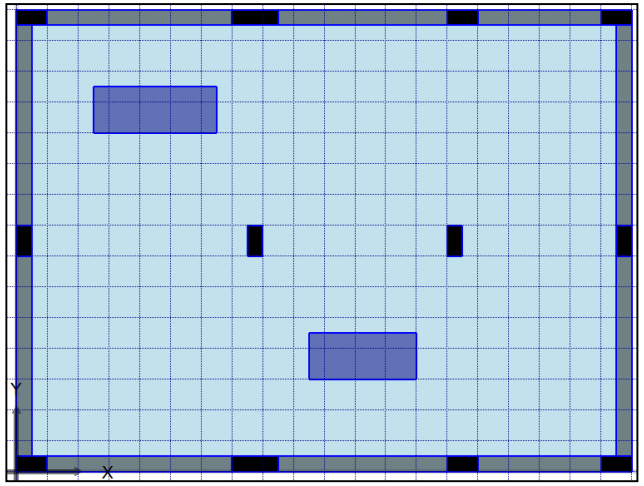
www.be4e.com

2. Build model and define its elements

The user can insert the model by three ways

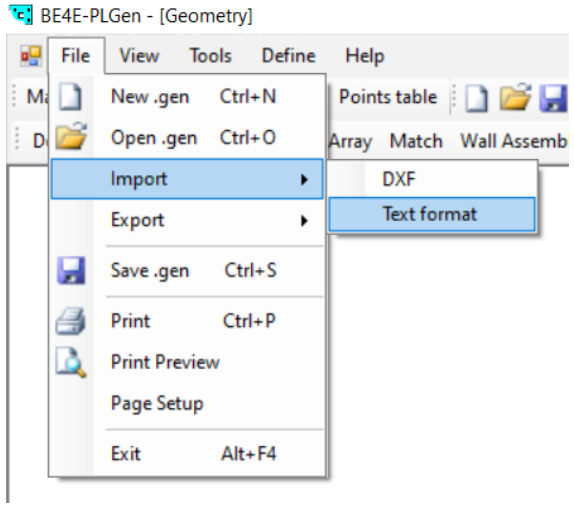
Drawing Model from PLGen

This option is useful for small models.



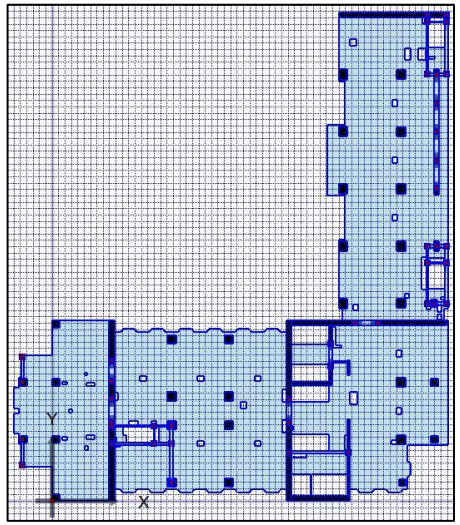
Importing Model from text files

This option is useful for developers.



Importing (.Dxf) file (AutoCAD file)

This option is useful for large models or detailed models.

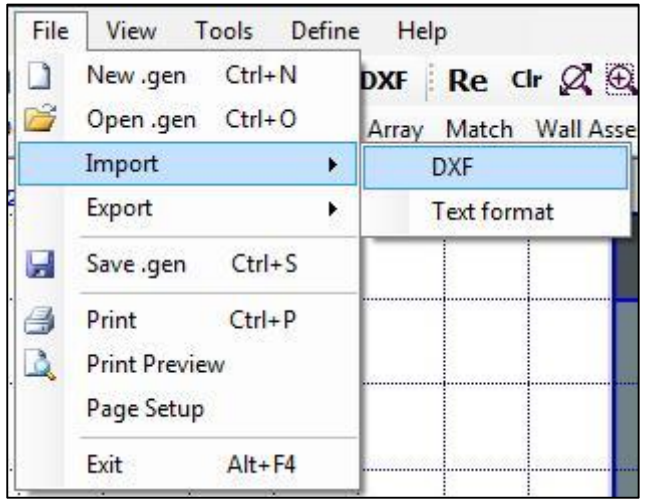


CUFE-BE

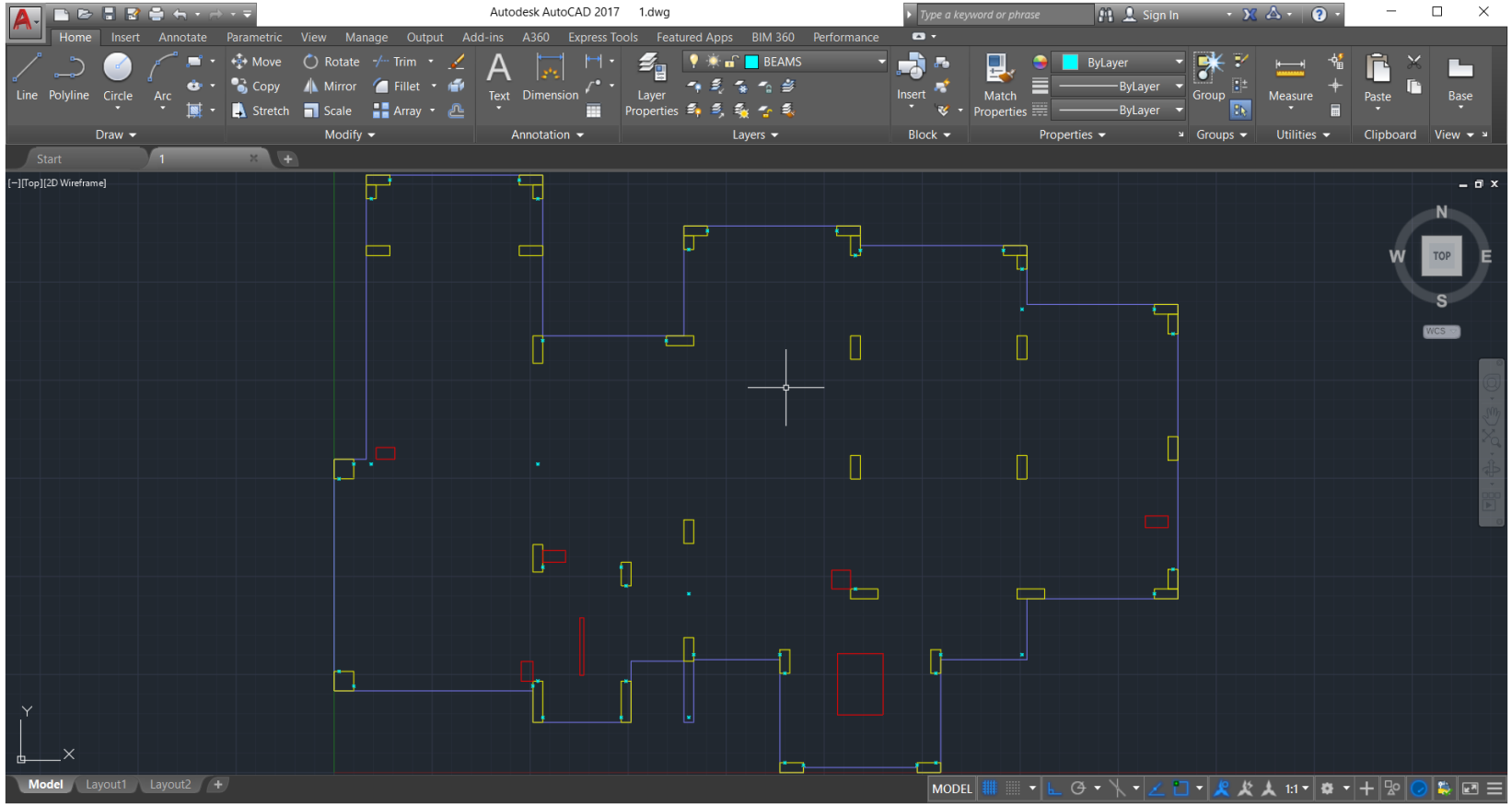
www.be4e.com

2. Build model and define its elements

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

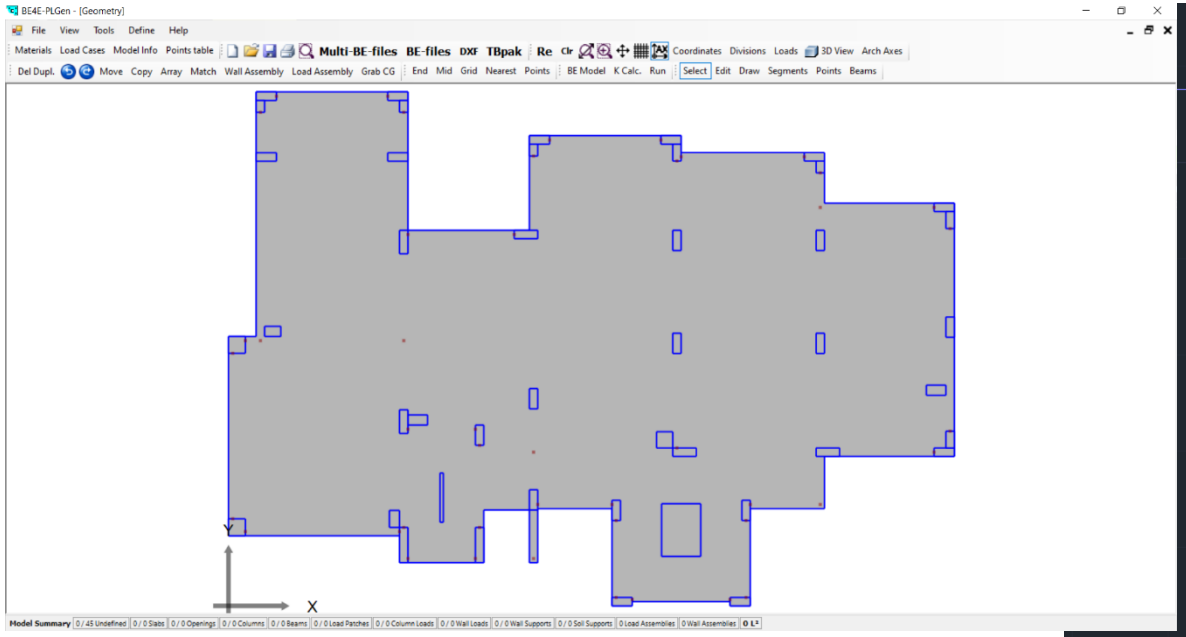


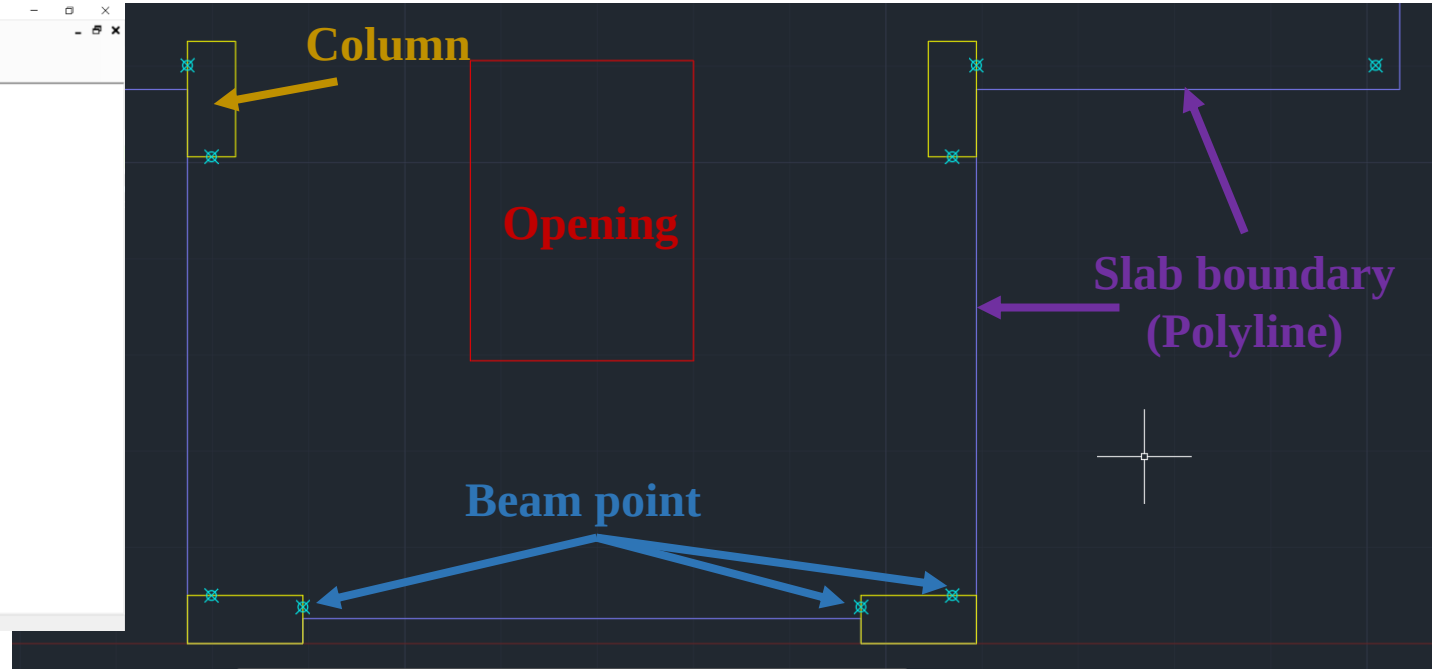
OR



2. Build model and define its elements

There are important notes that should be taken into consideration before Importing (.Dxf) file:
1- The slab line should be the outer line for the drawing.
2- All structural elements should be polyline in AutoCAD.
3- Beams should be inserted in AutoCAD is points at start/end of the beams.
4- All structural elements are drawn by four points only except the slab and the opening.





CUFE-BE

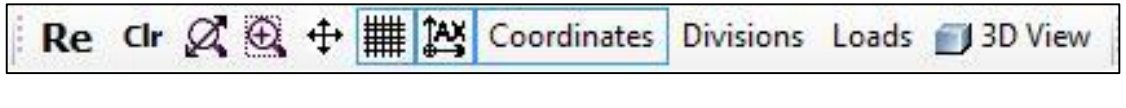
www.be4e.com

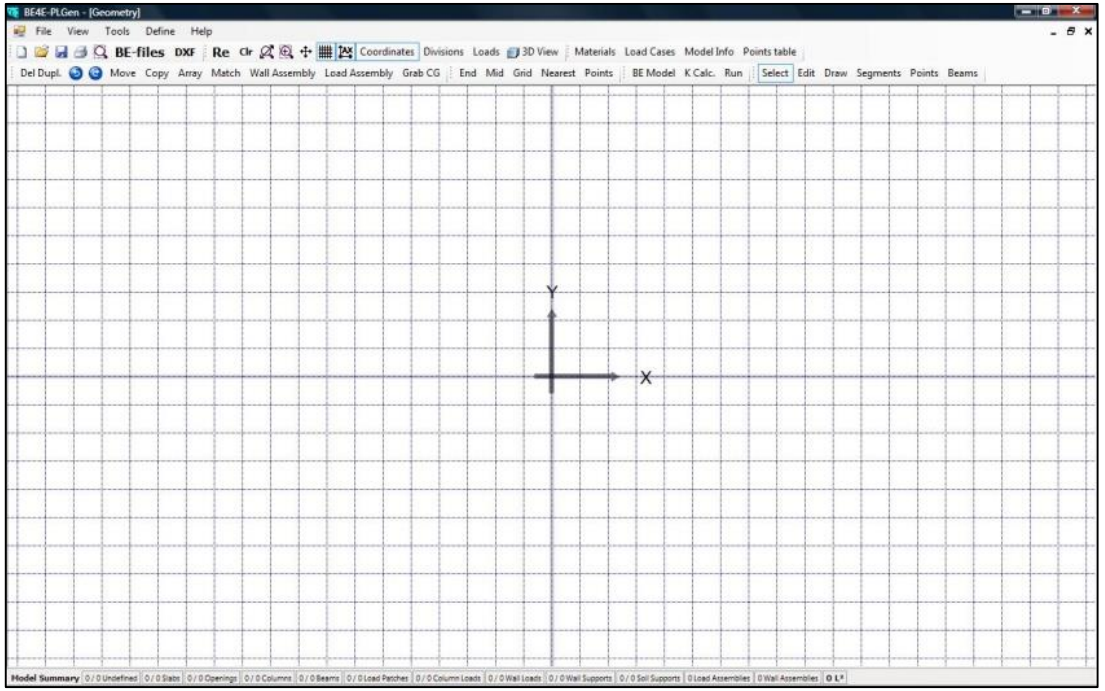
2. Build model and define its elements

B- Drawing Model from PLGen.

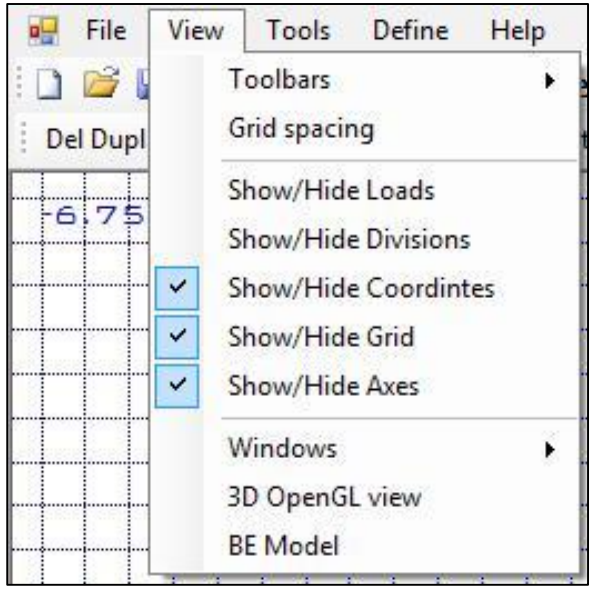
1- The user should show the Grids and show Coordinates.

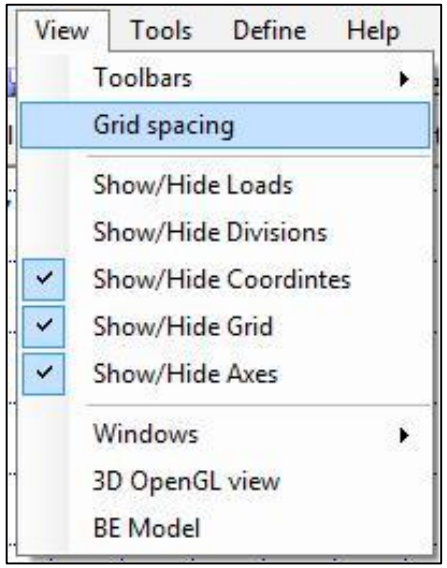
The user can change the Grid spacing from view menu.

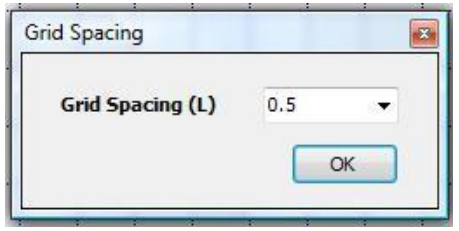




OR





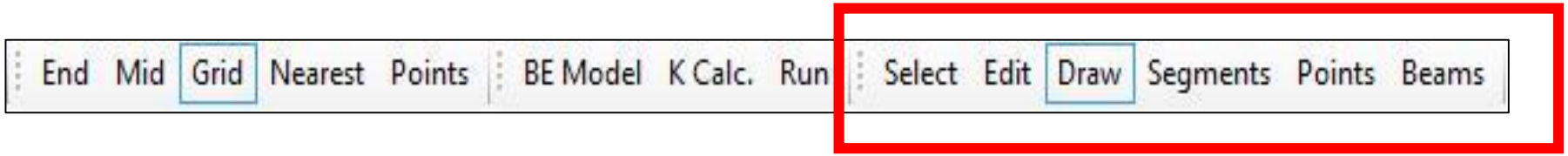


2. Build model and define its elements

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.



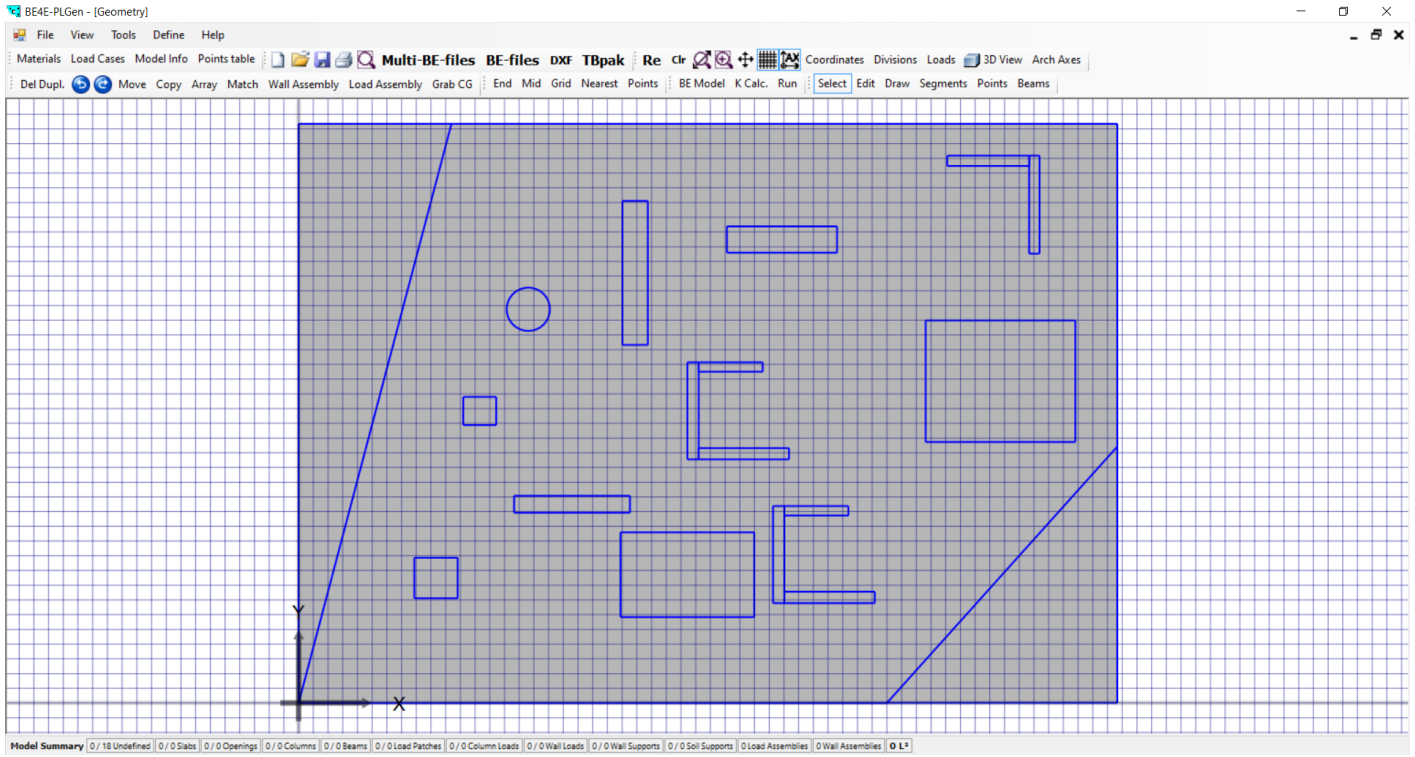
PLGen modes

2. Build model and define its elements

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

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.

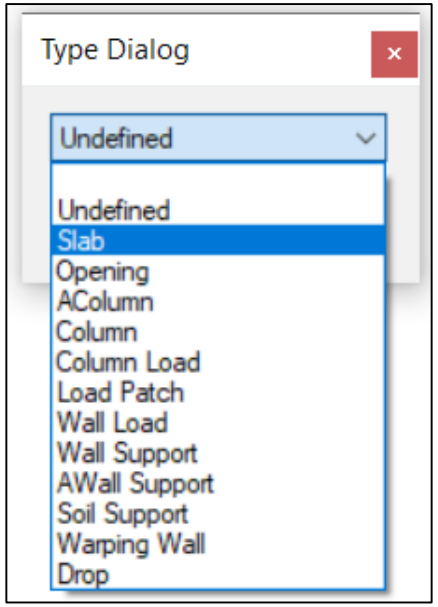
It has to be noted that, Acolumn, Awall support, Warping wall are special elements used in fixed base package.

- Loading elements. → Drawn using 4 point only.

Column load, Load patch, Wall load.

Defining different objects:

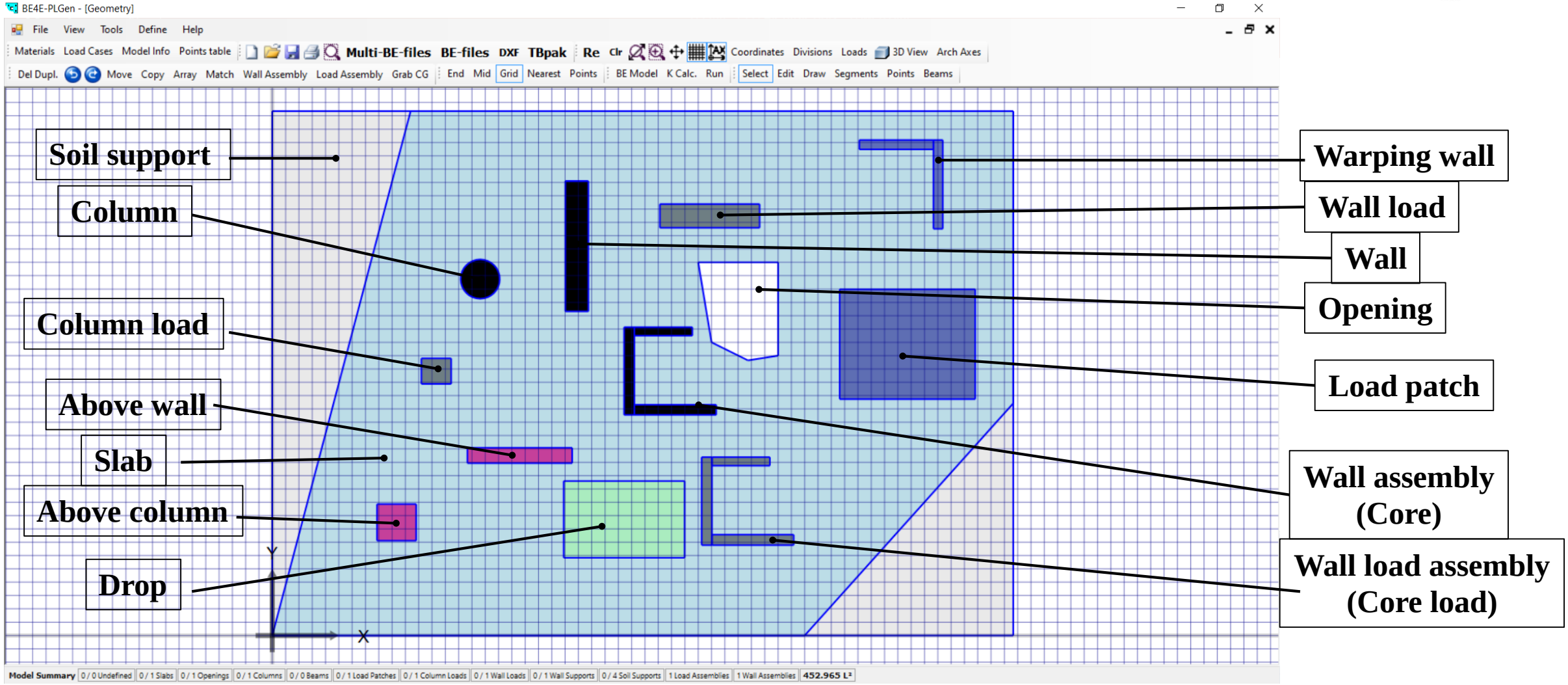
- Choose “select mode”.
- Click left click on the object to select it.
- Click right click to choose object type.



CUFE-BE

www.be4e.com

2. Build model and define its elements

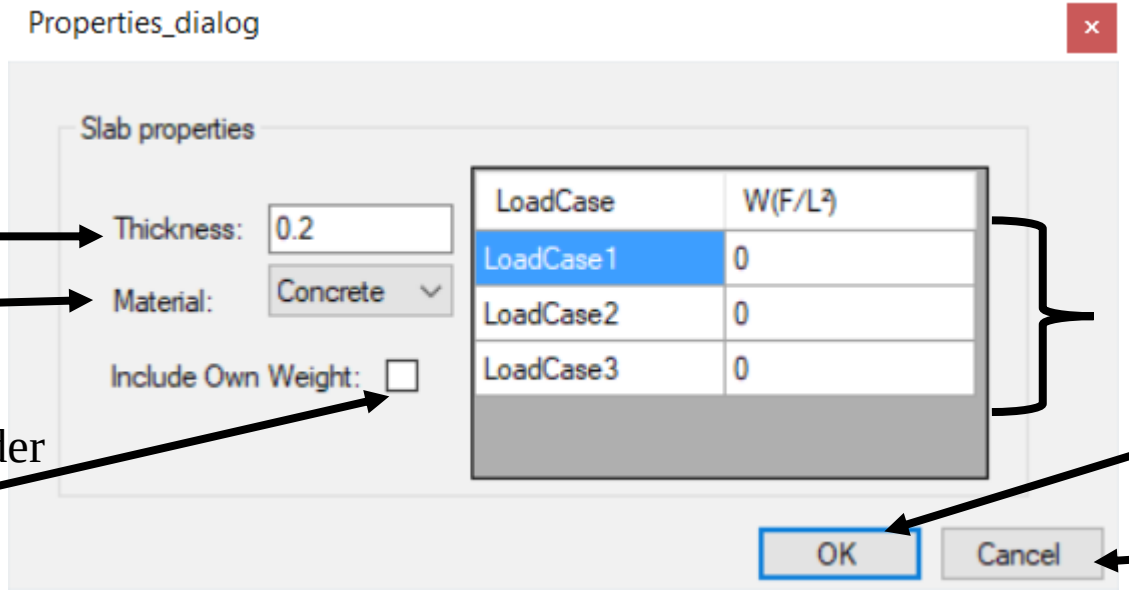


2. Build model and define its elements

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

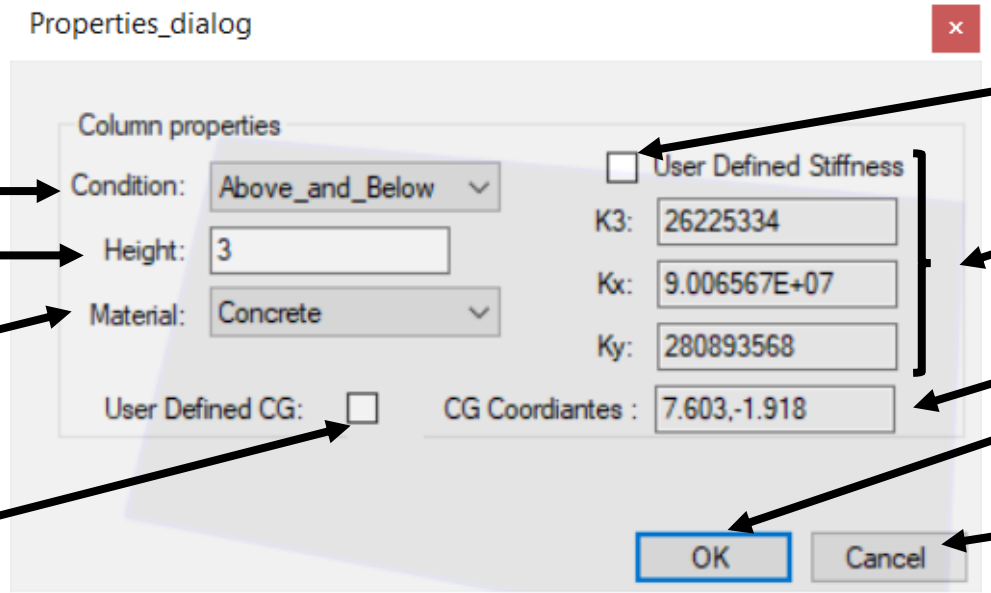
Cancel button to dismiss changes in the slab data

2. Build model and define its elements

<u>Supporting elements</u> (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

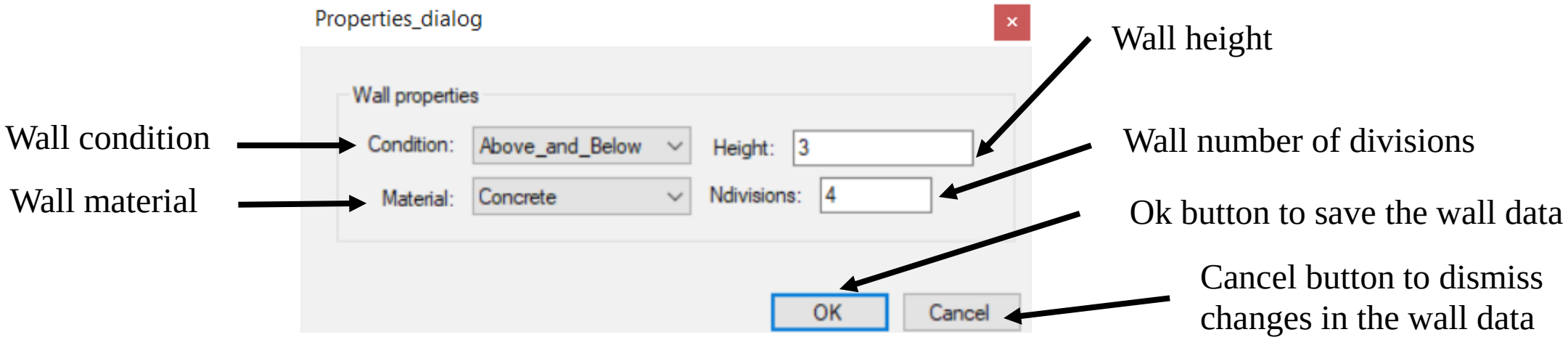
Cancel button to dismiss changes in the column data

2. Build model and define its elements

<u>Supporting elements</u> (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.

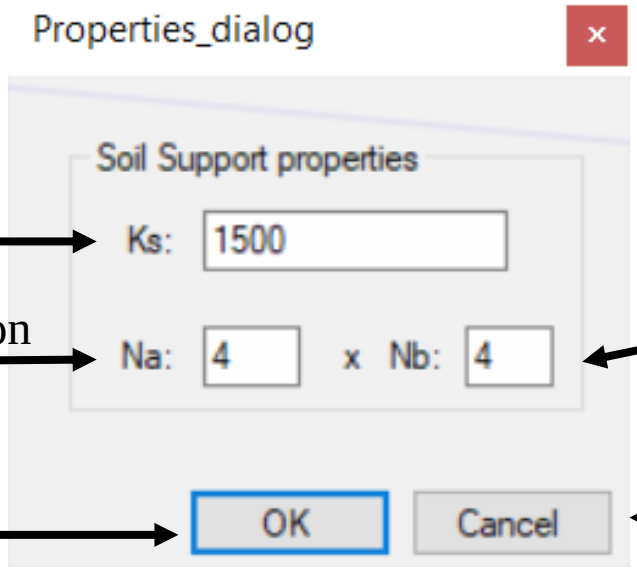


2. Build model and define its elements

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

Soil number of divisions in ‘a’ direction

Soil number of divisions in ‘b’ direction

Ok button to save the soil data

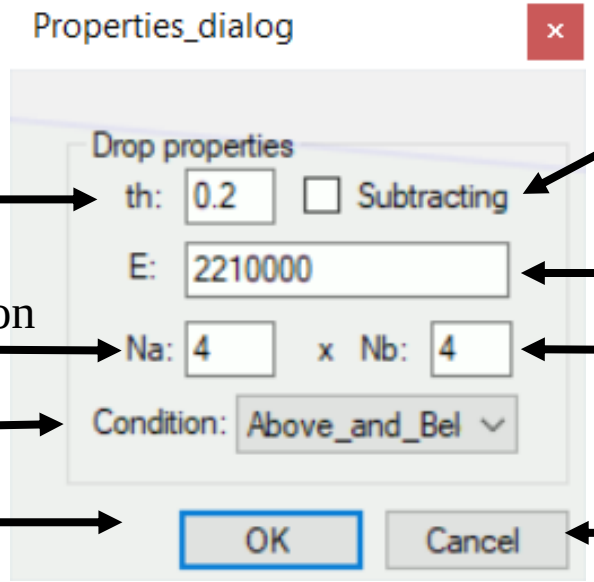
Cancel button to dismiss changes in the soil data

2. Build model and define its elements

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

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

Drop modulus of elasticity

Drop number of divisions in ‘a’ direction

Drop number of divisions in ‘b’ direction

Drop condition

Ok button to save the drop data

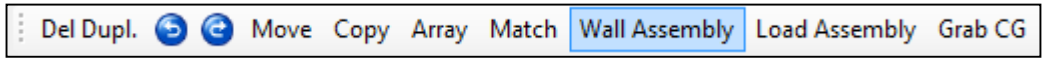
Cancel button to dismiss changes in the drop data

2. Build model and define its elements

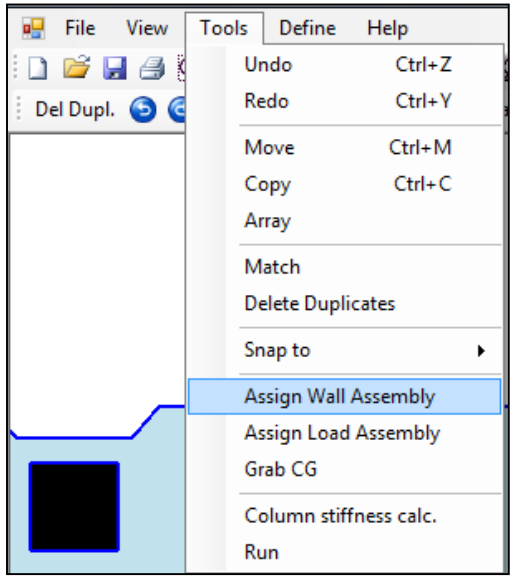
Supporting elements (Wall assembly)

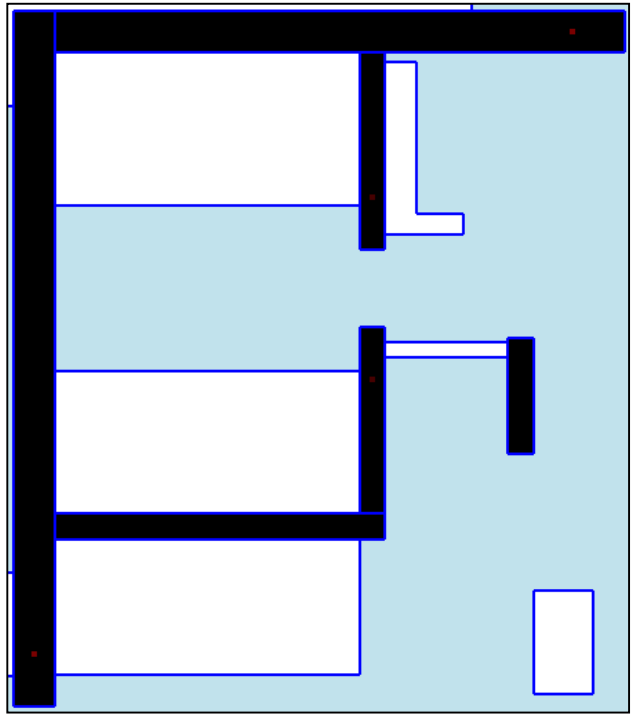
Assemblies is used if the user want to combine either support together like cores or any shape of shear walls etc.
1- Assign wall assembly (core) elements as wall support.
2- Select all walls that need to be combined then click on wall assembly button.

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



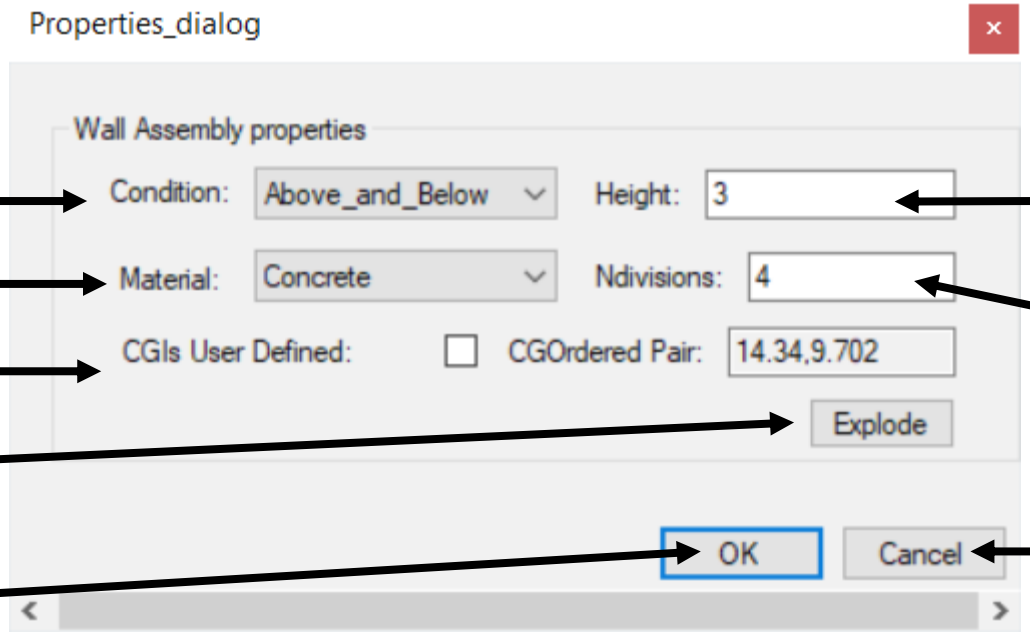


2. Build model and define its elements

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

Wall assembly material

Wall assembly CG

Explode tab to restore the shear walls again

Ok button to save the wall assembly data

Wall assembly height

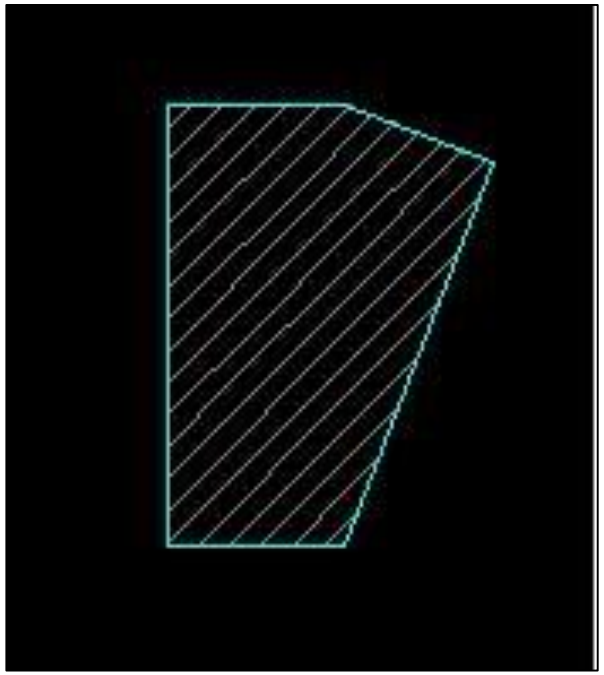
Wall assembly number of division per each wall

Cancel button to dismiss changes in the wall assembly data

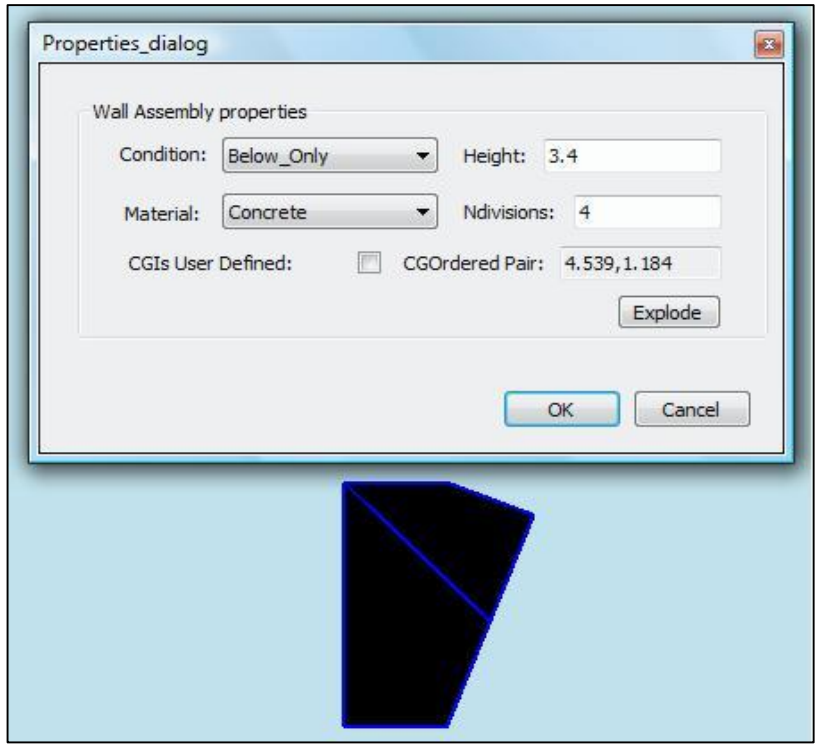
2. Build model and define its elements

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

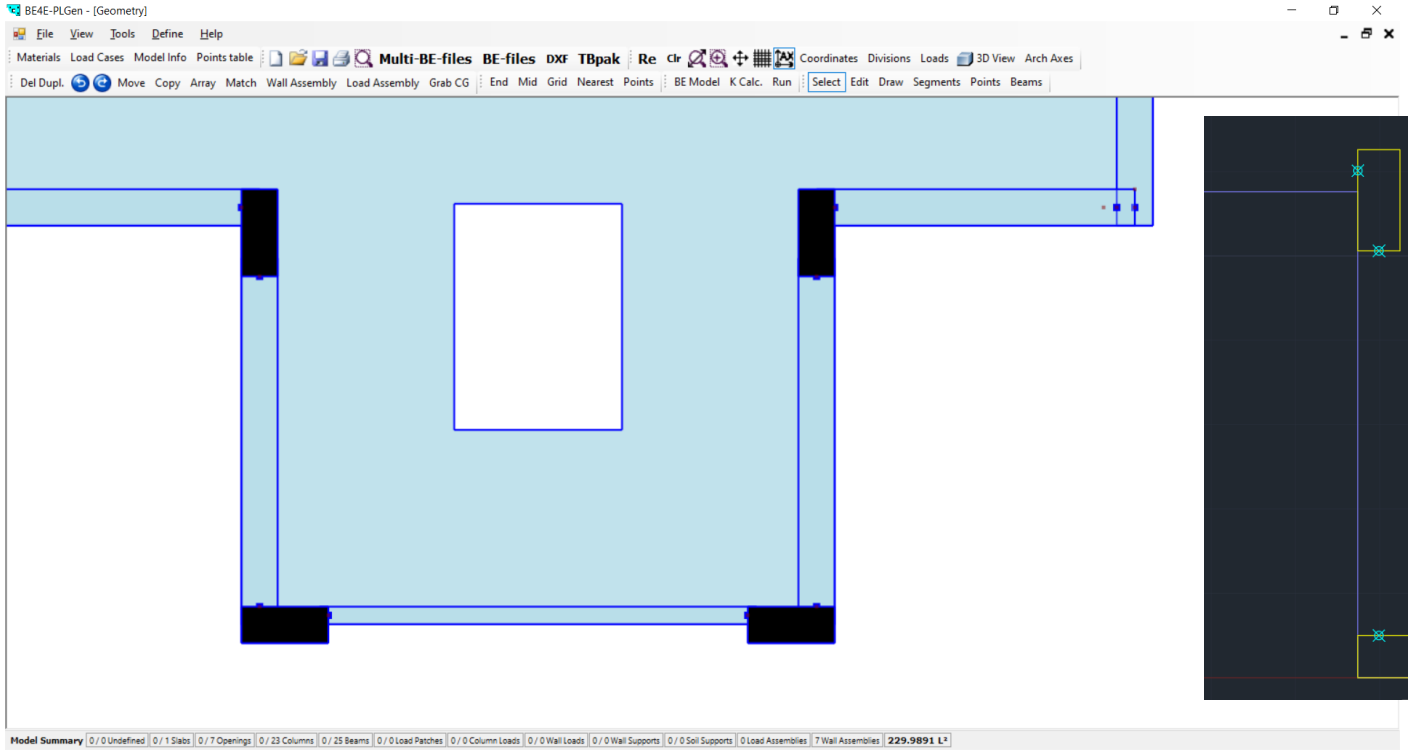


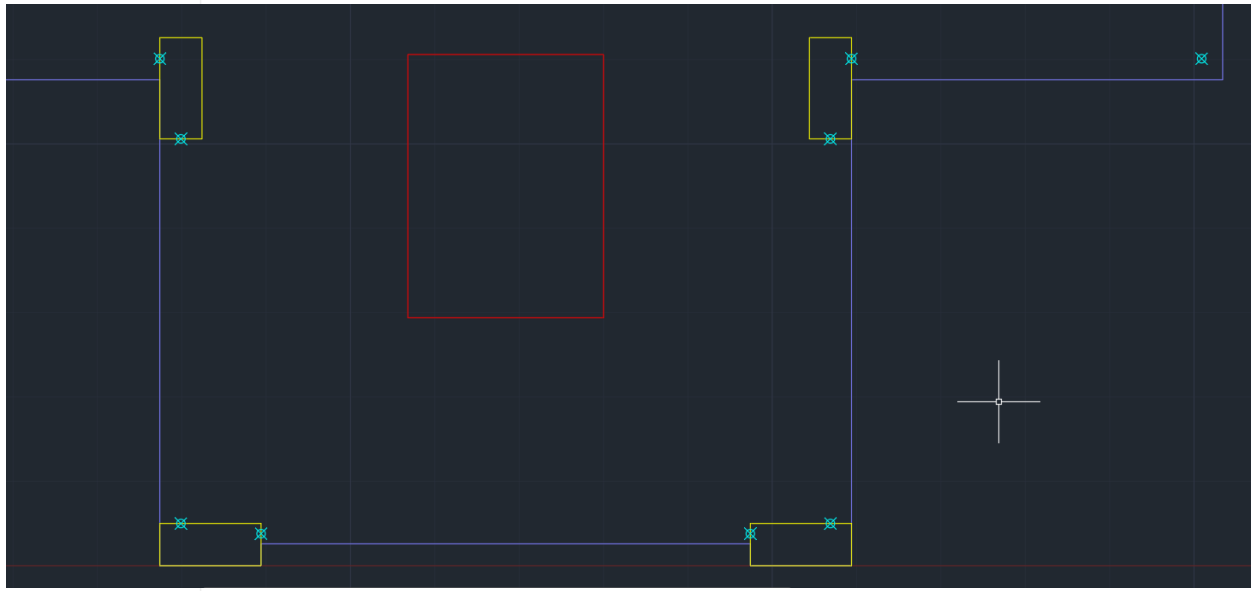
Non-quadratic column in PLGen

2. Build model and define its elements

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.



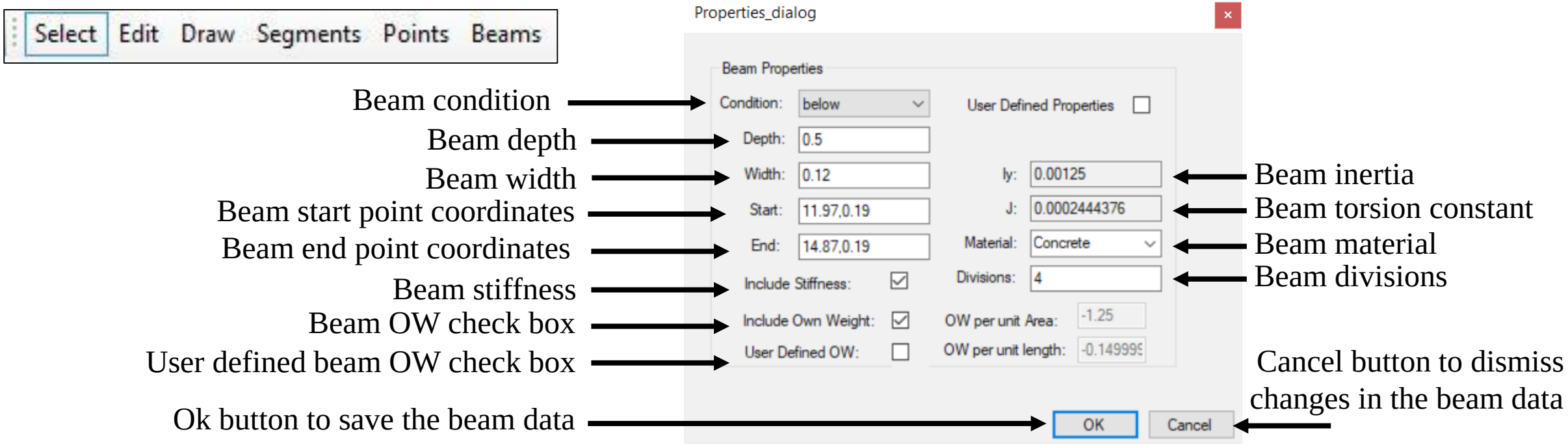


2. Build model and define its elements

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).



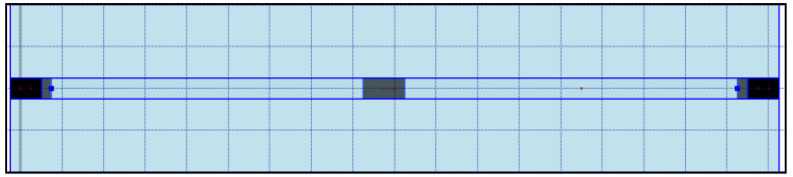
2. Build model and define its elements

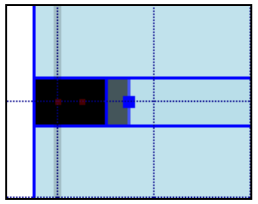
Supporting elements (Beams)

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.

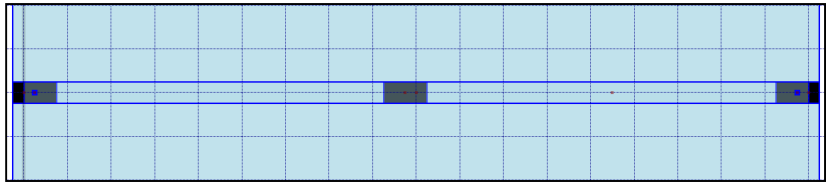


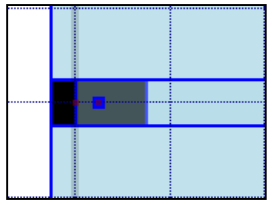


$L_{in} \leq 0.5\ B_{width}$

In case of fixed beams

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





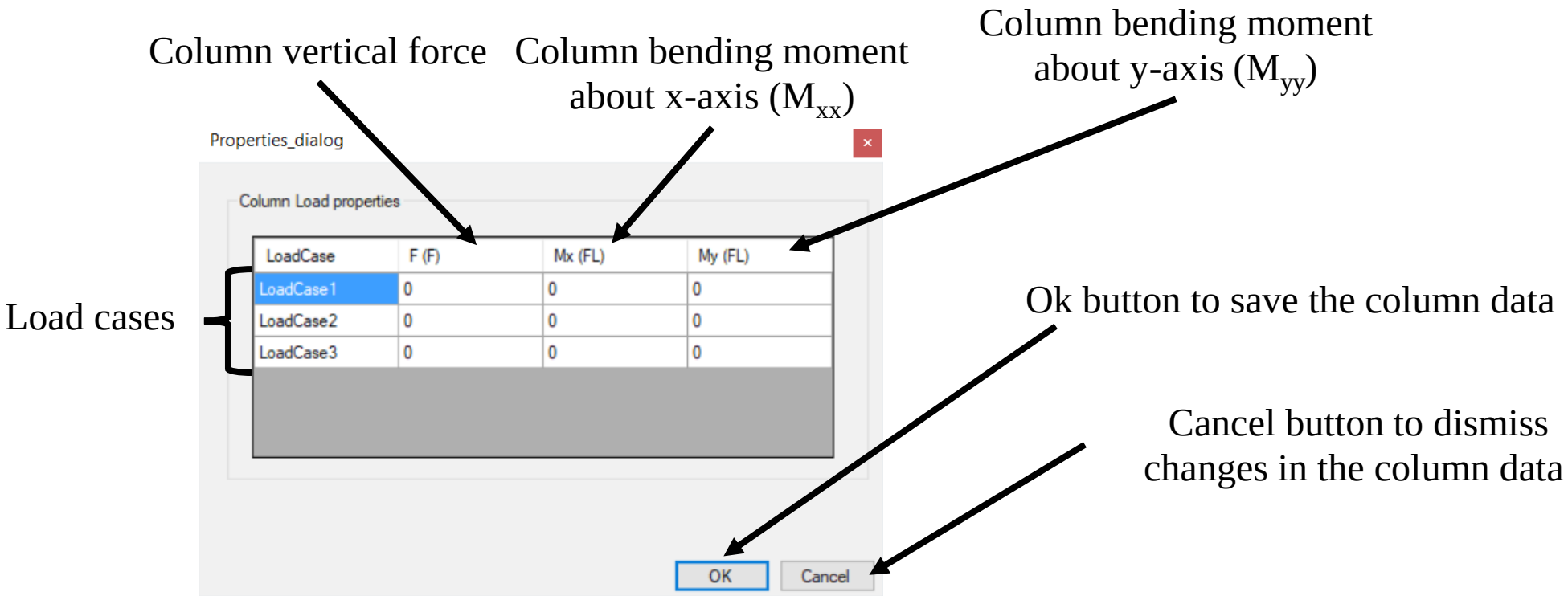
$L_{in} \geq 0.5\ B_{width}$

2. Build model and define its elements

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.



2. Build model and define its elements

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.

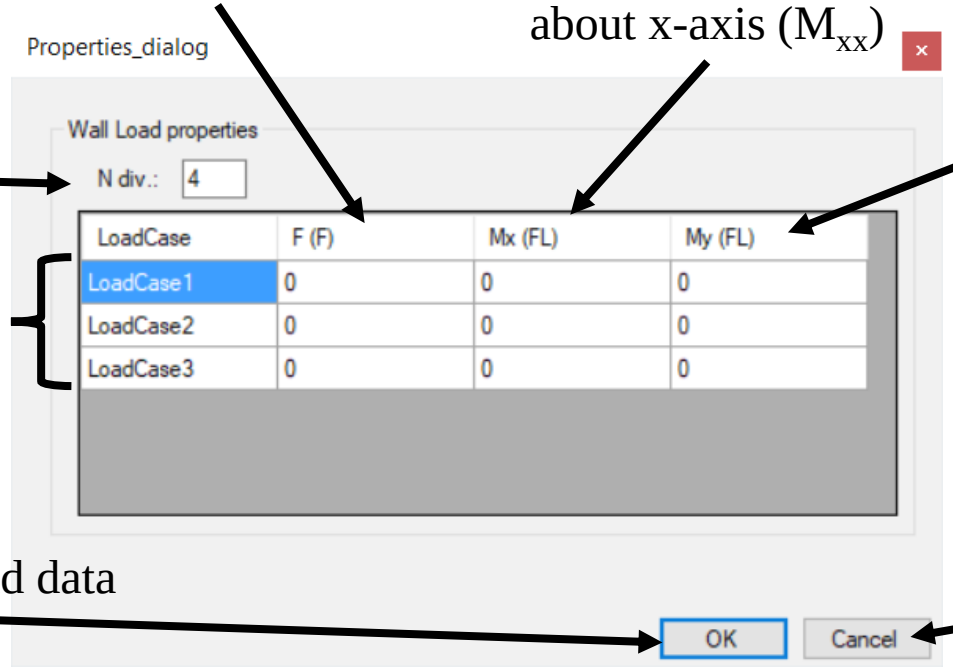
Wall load vertical force

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



Ok button to save the wall load data

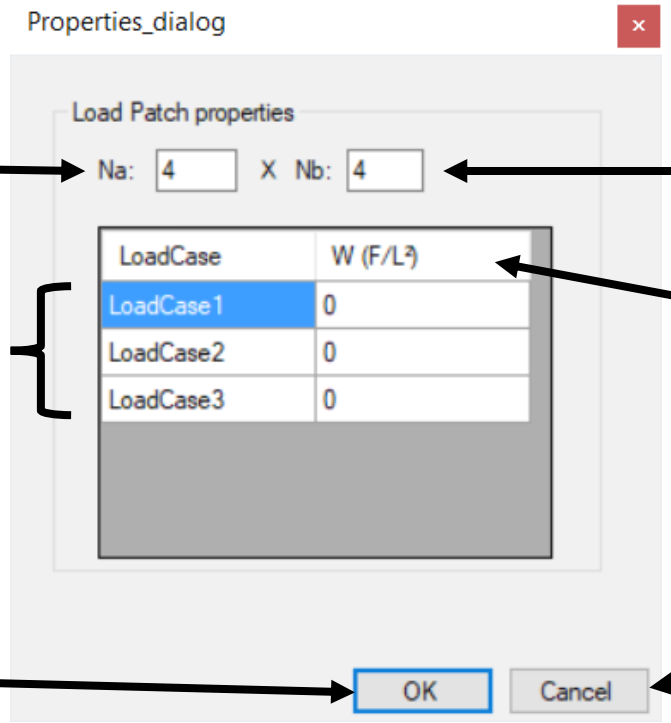
Cancel button to dismiss changes in the wall load data

2. Build model and define its elements

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.



Load patch number of divisions in ‘a’ direction

Load patch number of divisions in ‘b’ direction

Load cases

Uniform load value

Ok button to save the load patch data

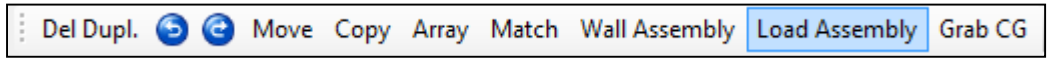
Cancel button to dismiss changes in the load patch data

2. Build model and define its elements

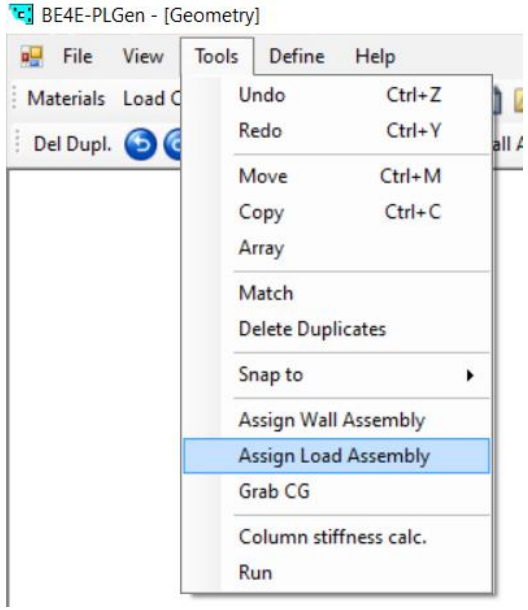
Loading elements (Wall load assembly)

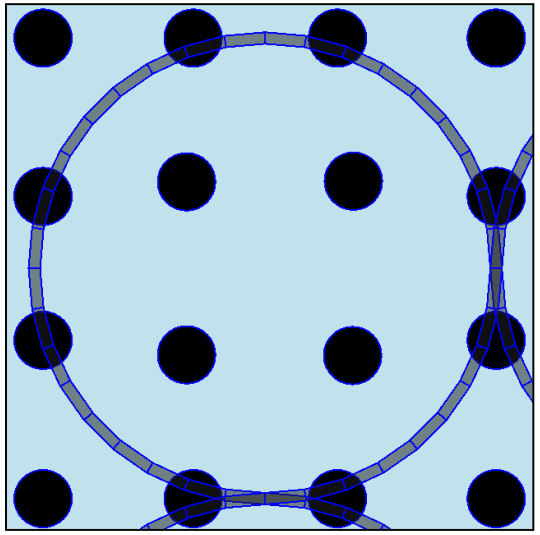
Load assemblies is used if the user want to combine either load together like core's load or any shape of shear wall's load.
1- Assign wall load assembly (core) elements as wall load.
2- Select all wall loads that need to be combined then click on wall assembly button.
If the user press right click on 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





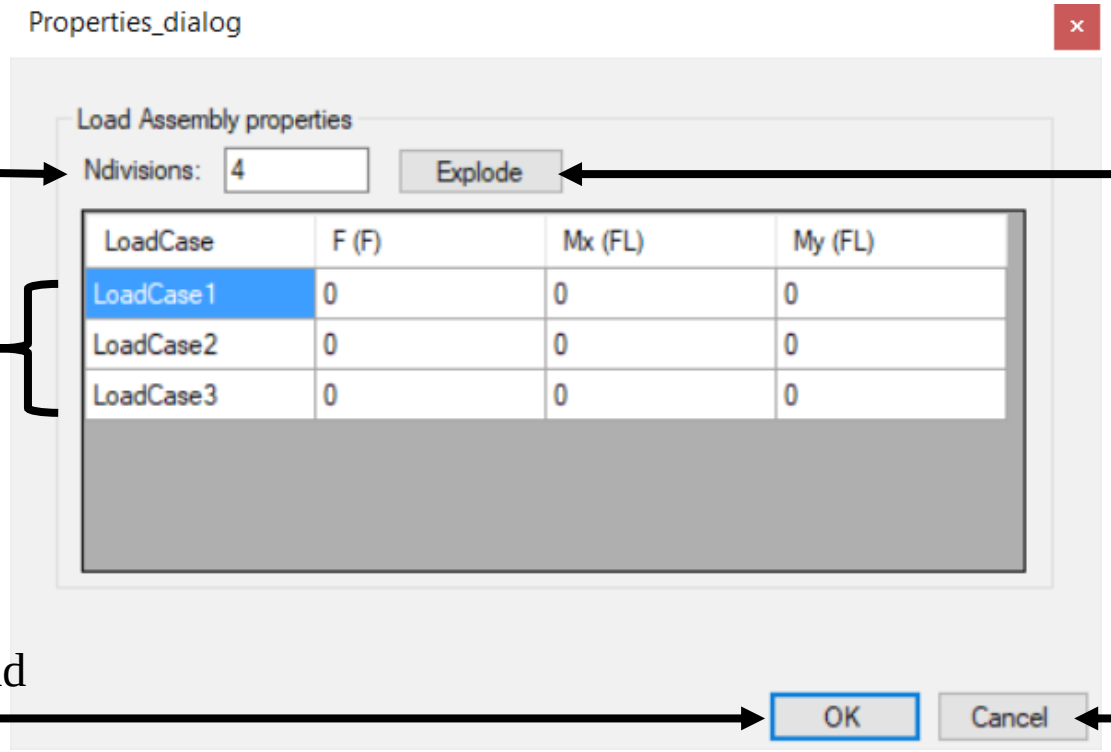
2. Build model and define its elements

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

Explode tab to restore the wall loads again



Load cases

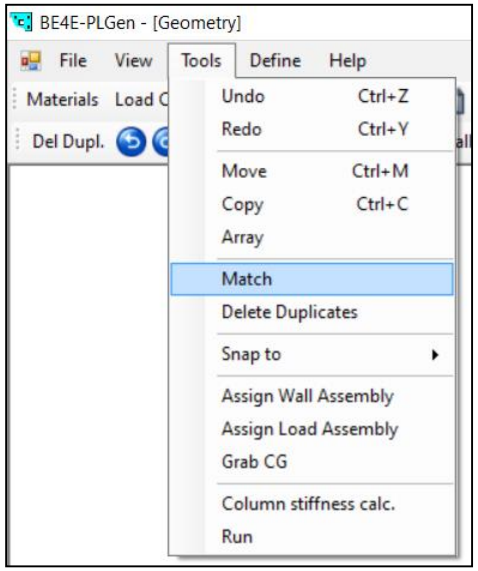
Ok button to save the wall load assembly data

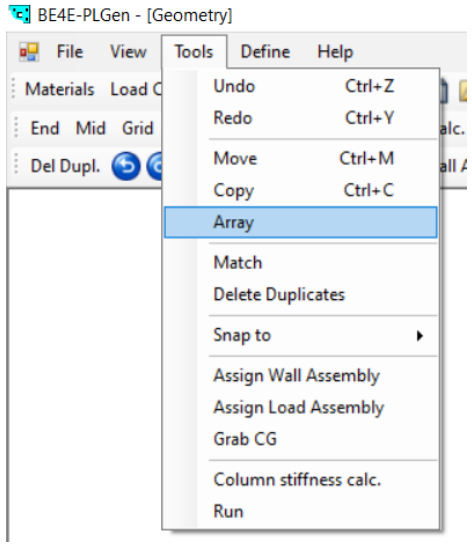
Cancel button to dismiss changes in the wall load assembly data

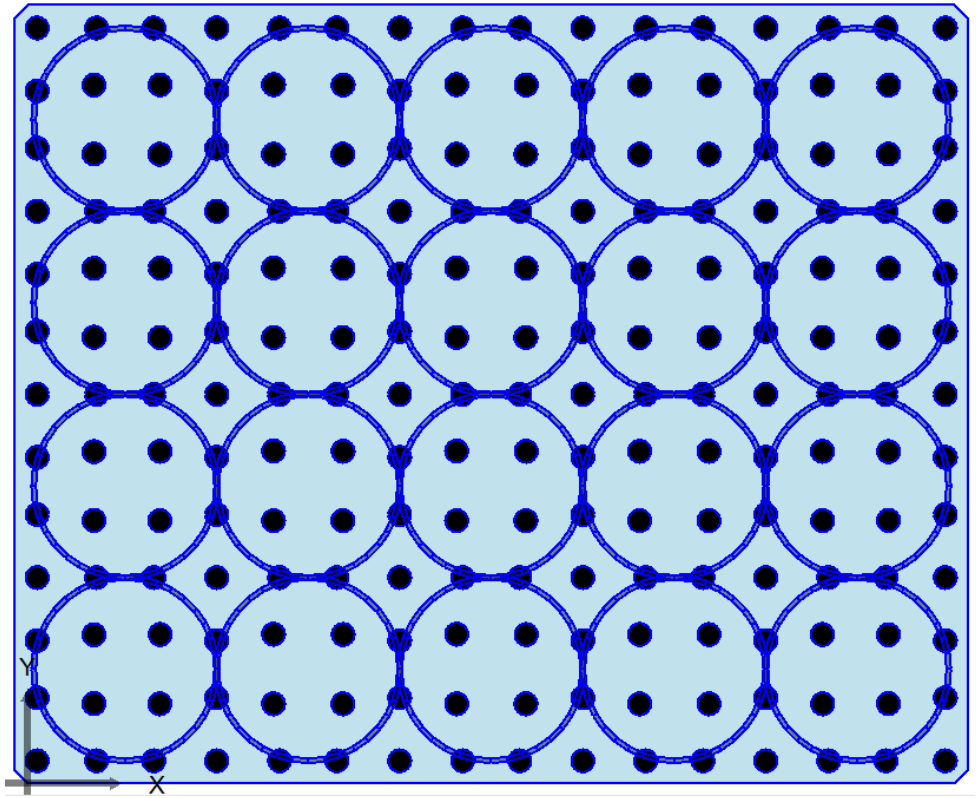
2. Build model and define its elements

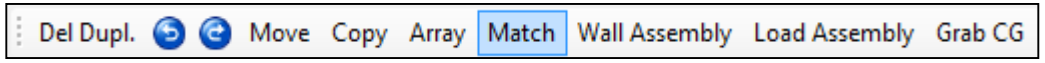
Other PLGen tools

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



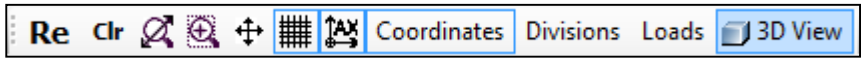




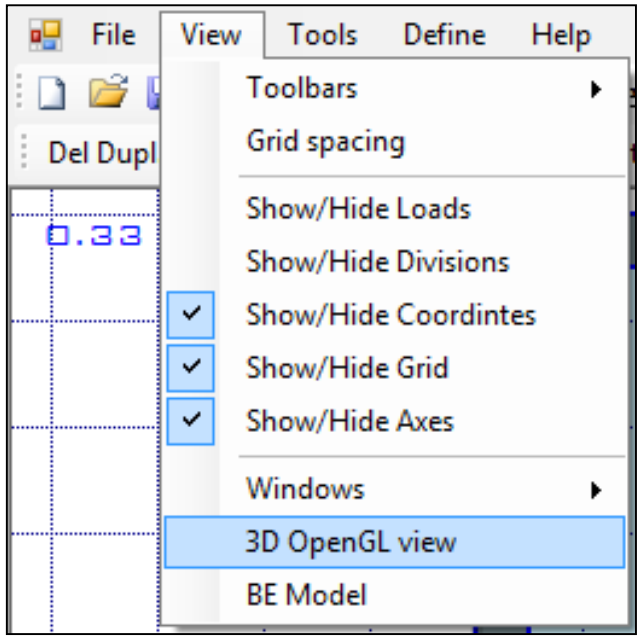


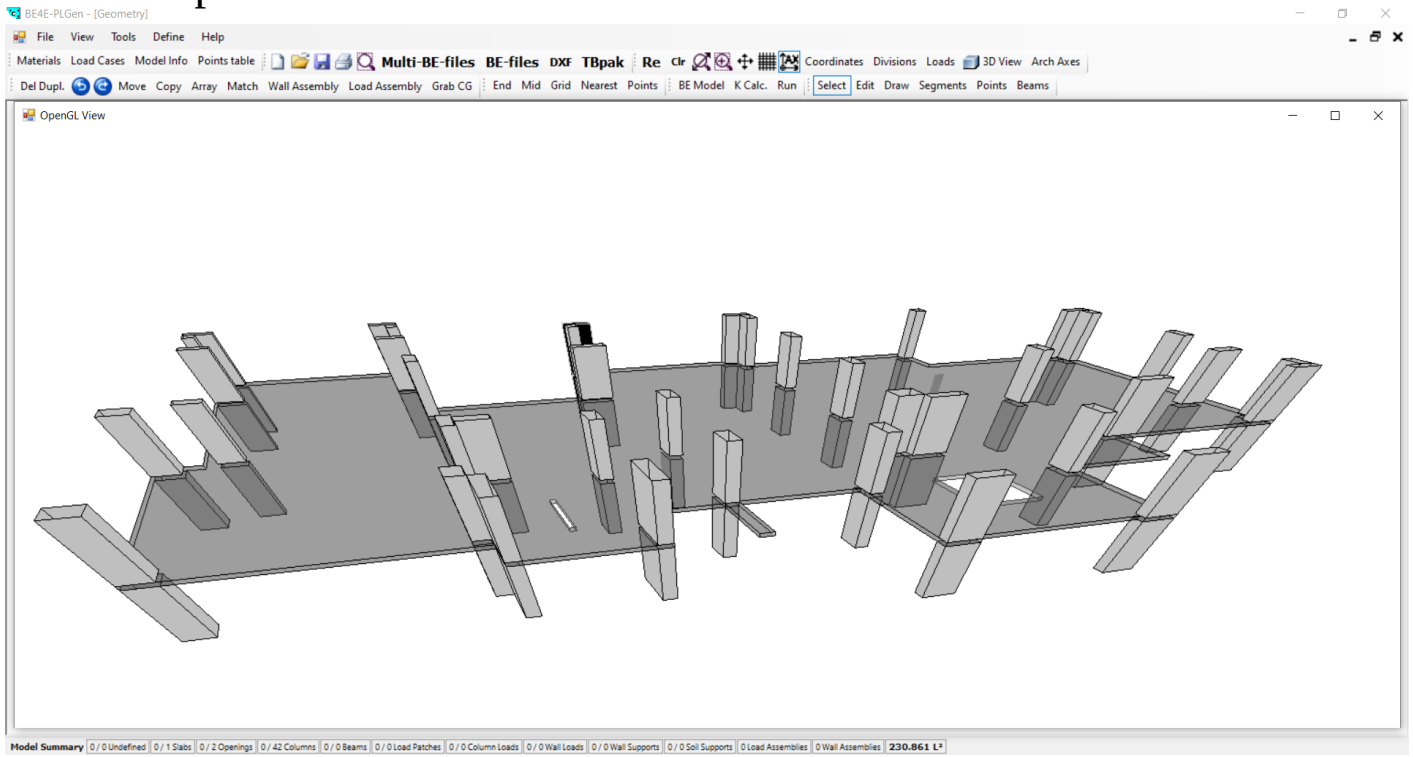
2. Build model and define its elements

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.





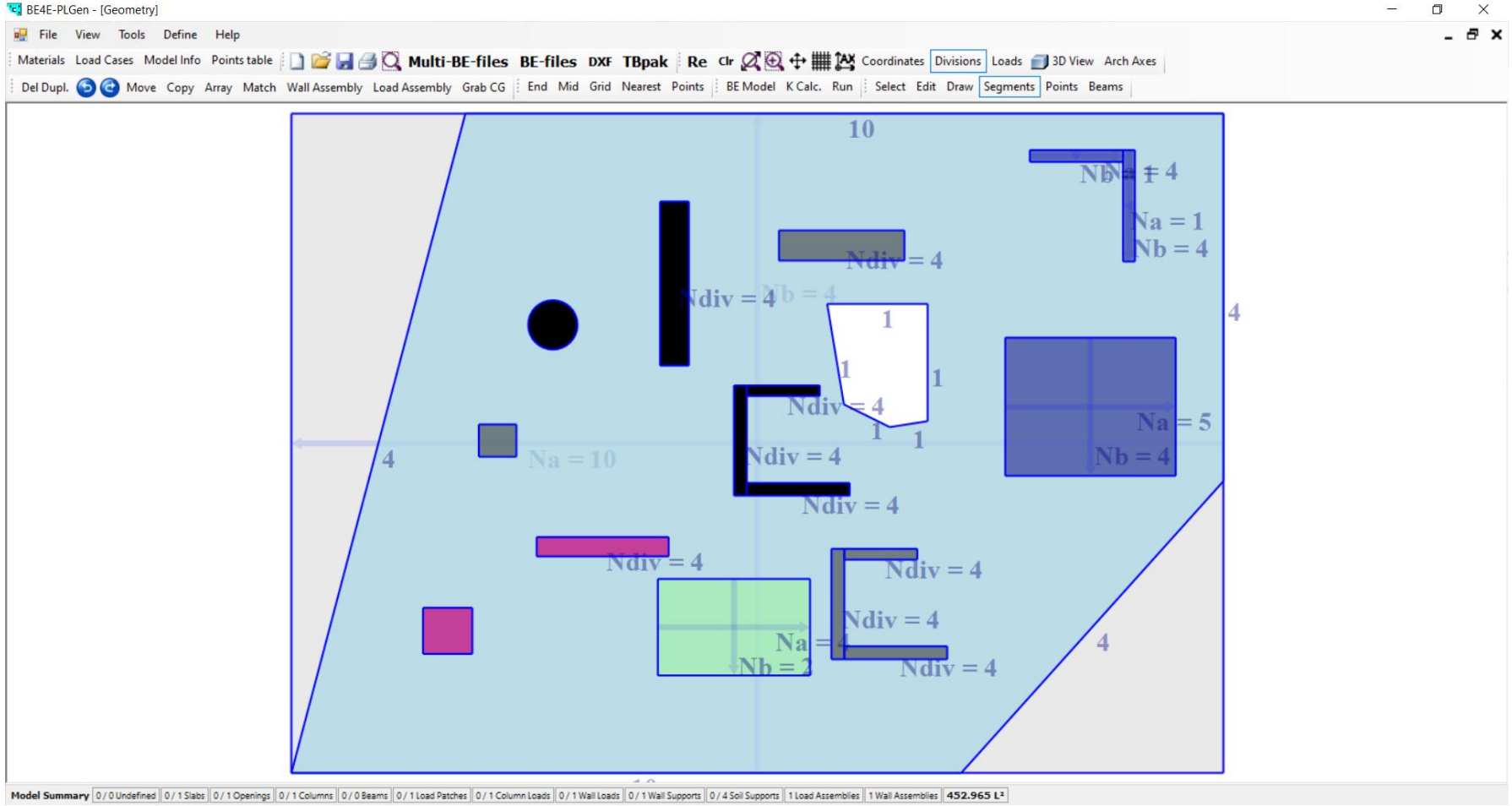
CUFE-BE

www.be4e.com

2. Build model and define its elements

Other PLGen tools

- User can view the number of division for each side of the slab or opening, and divisions of any elements.

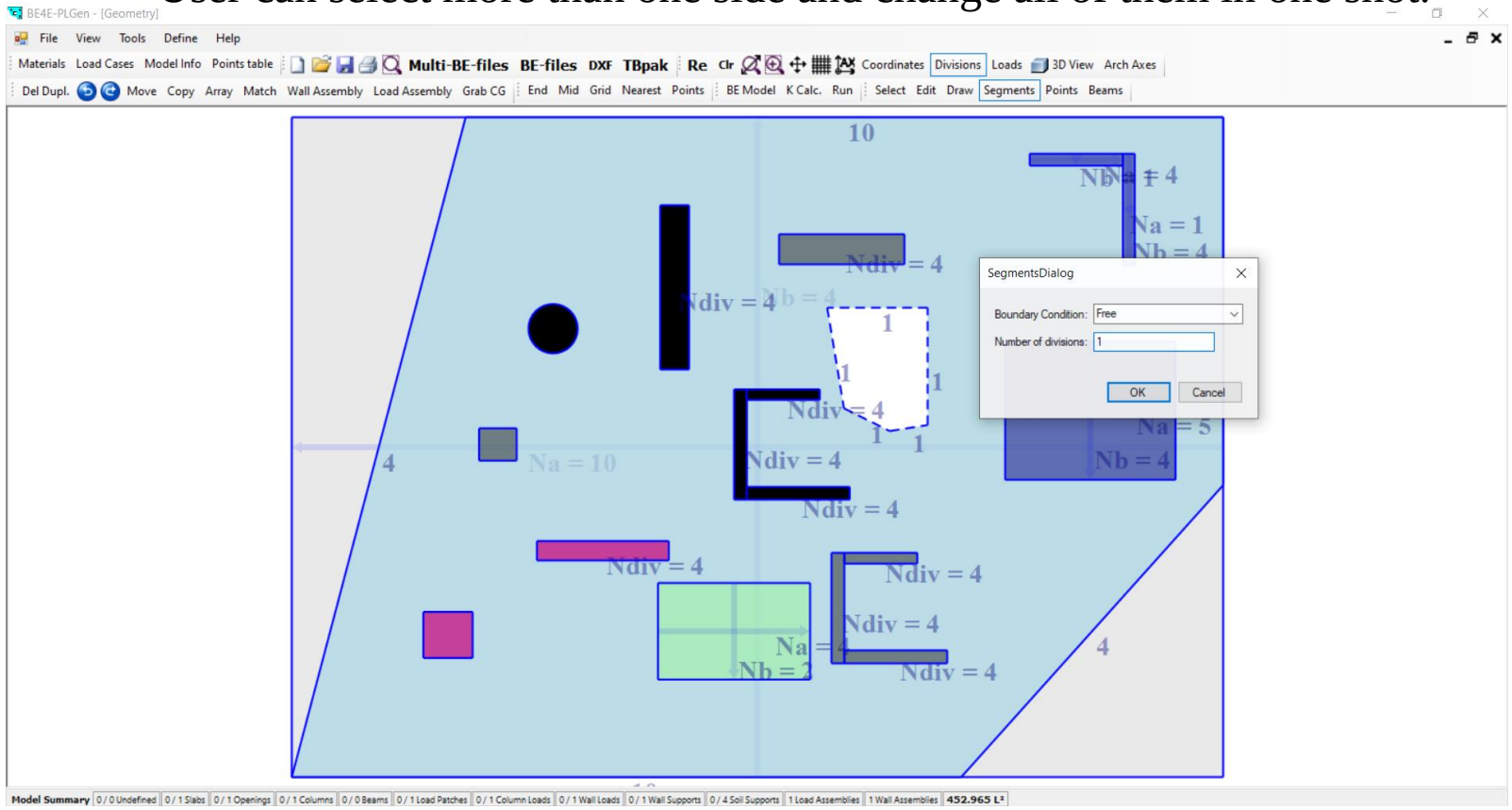


CUFE-BE

www.be4e.com

3. Edit the boundary element divisions

- 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.



CUFE-BE

www.be4e.com

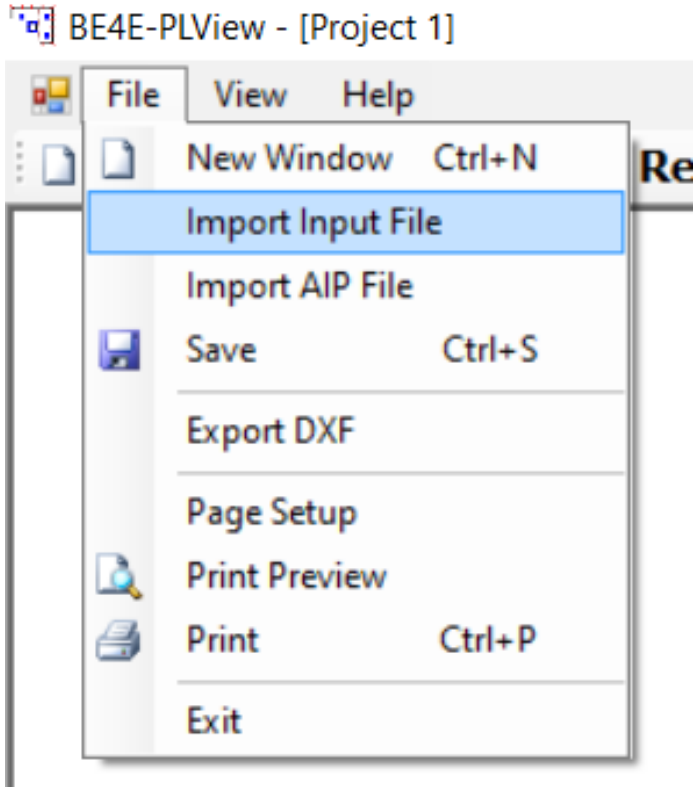
Table of contents

CUFE-BE

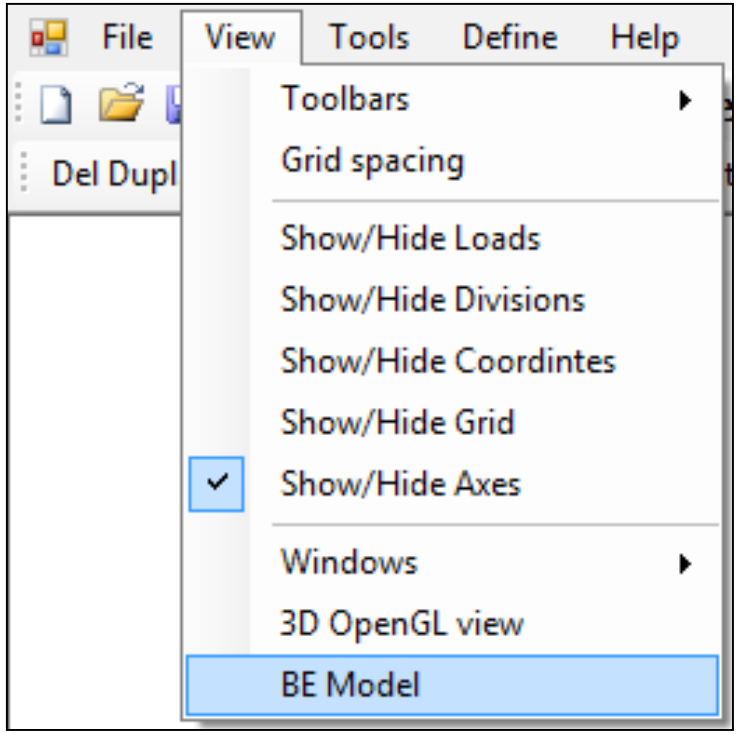
www.be4e.com

Table of contents

- 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



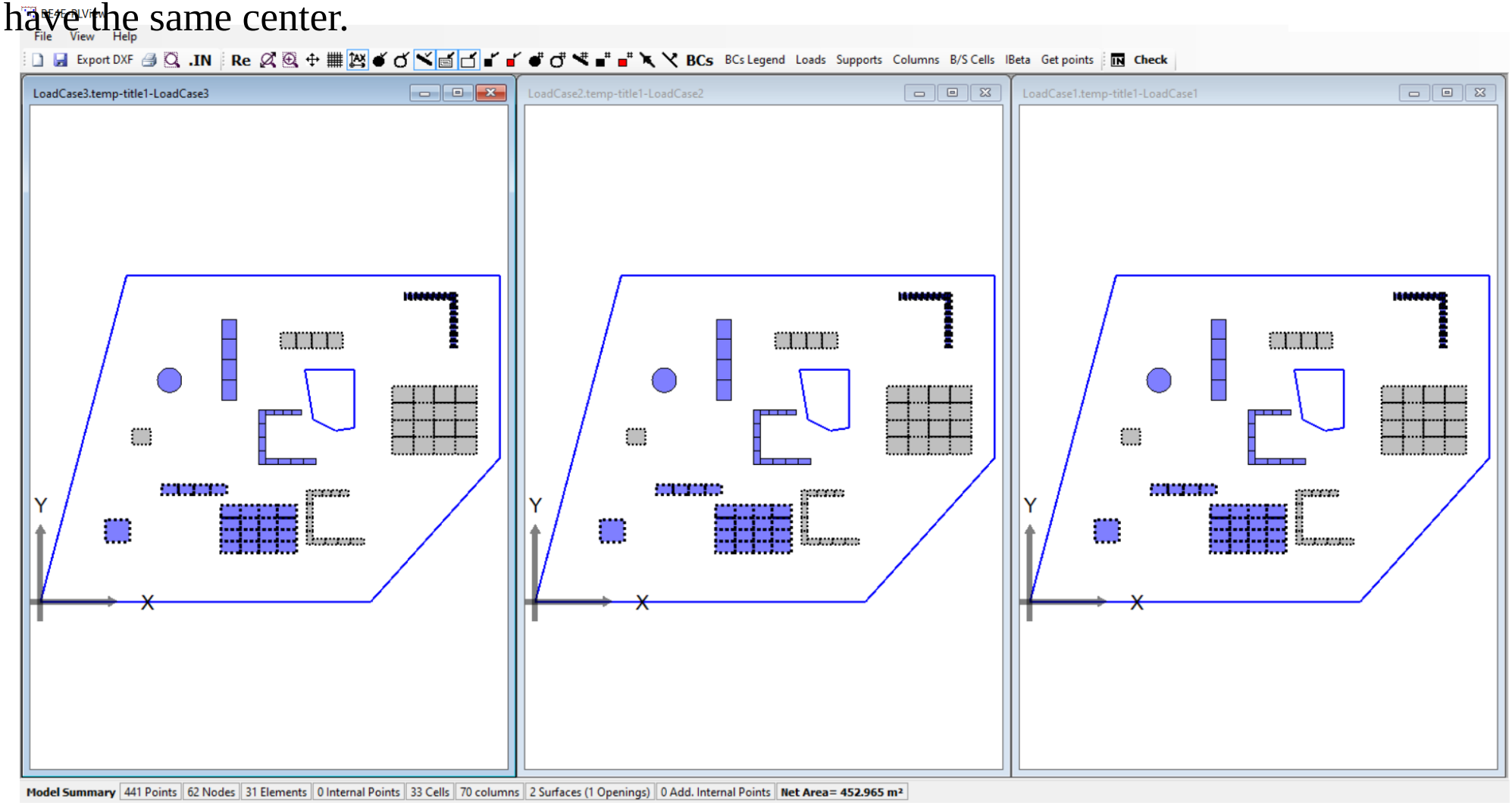
OR

From PLGen

CUFE-BE

www.be4e.com

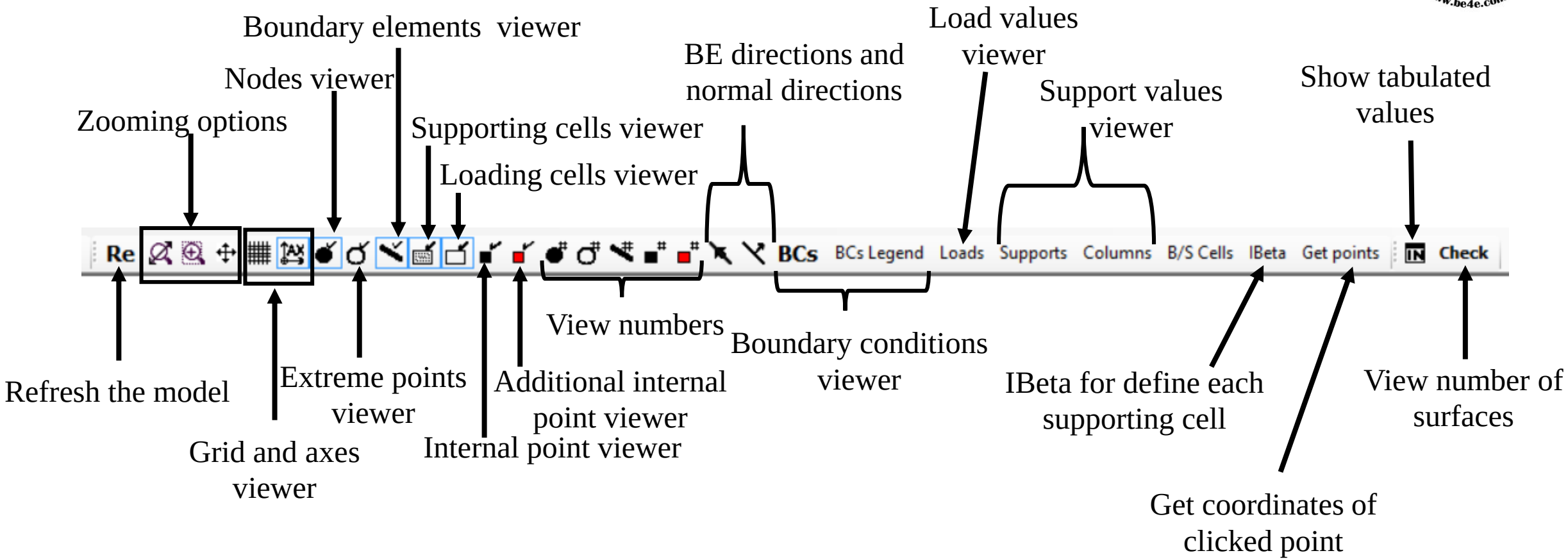
- PLView open window for each load case.
- The model will be unable to run in cases of, two node have the same coordinates or two supporting cells have the same center.



CUFE-BE

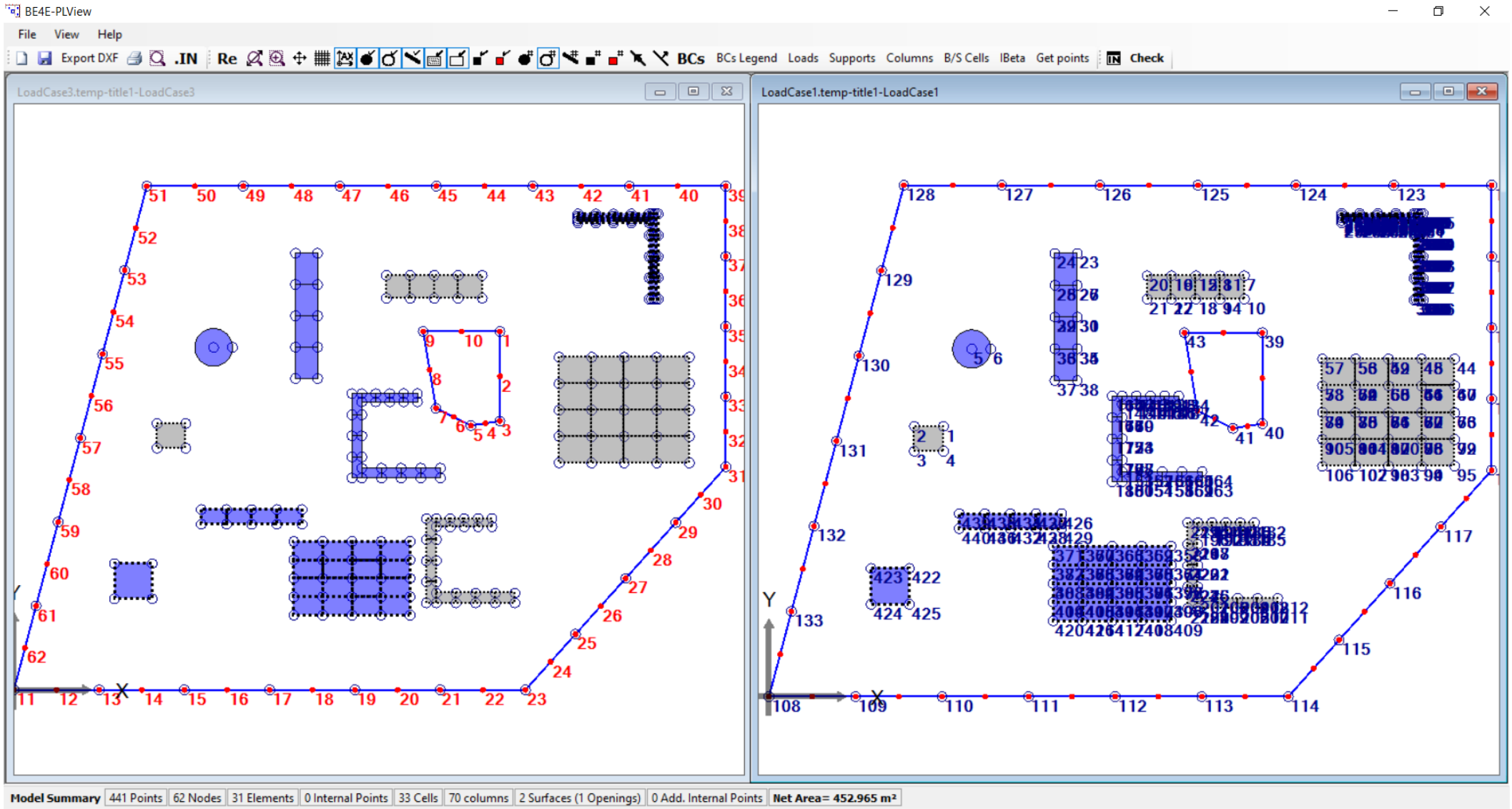
www.be4e.com

- In PLView user can use these features.



CUFE-BE

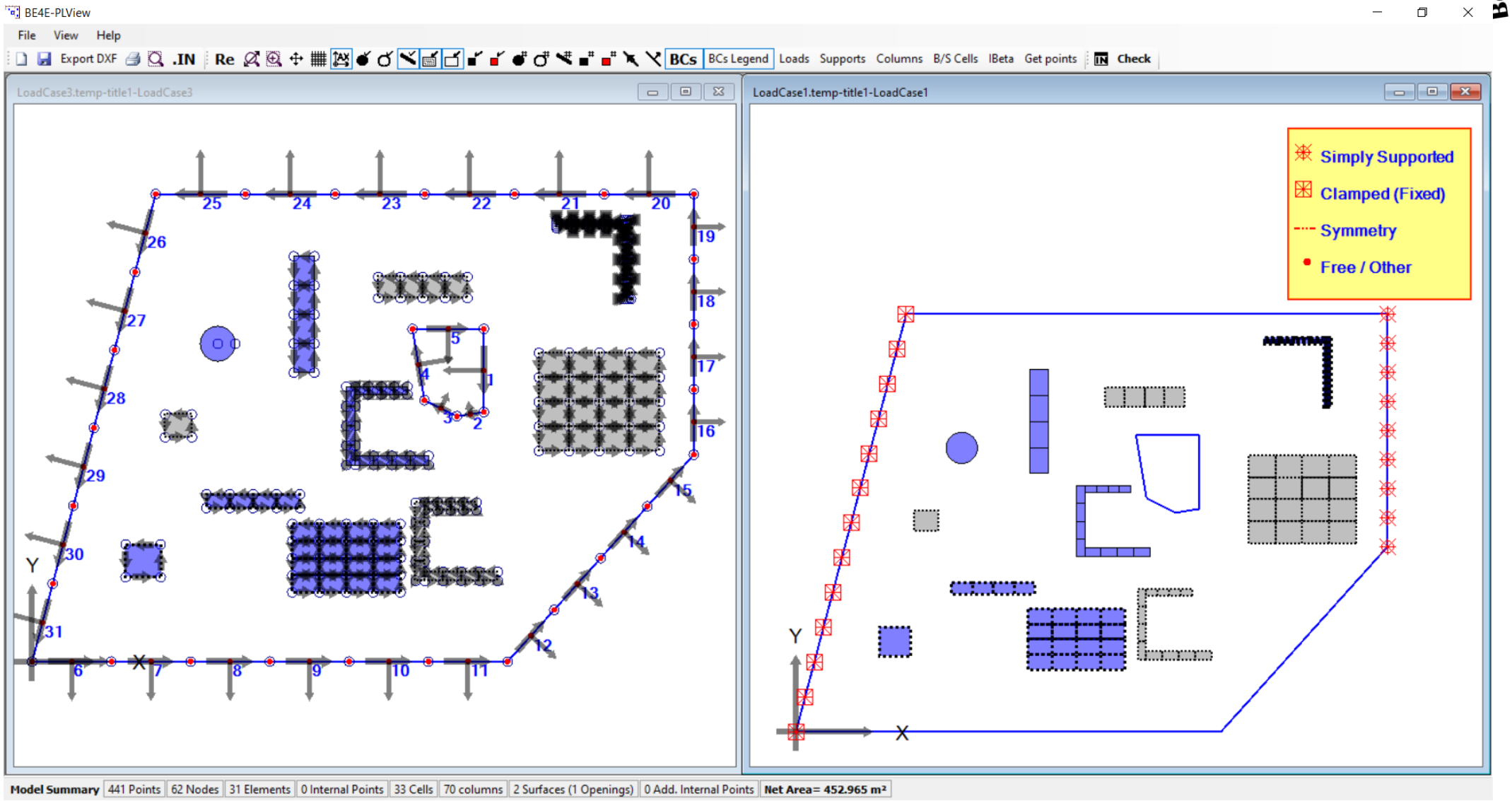
www.be4e.com



BE model of slab showing nodes, and extreme points numbers

CUFE-BE

www.be4e.com

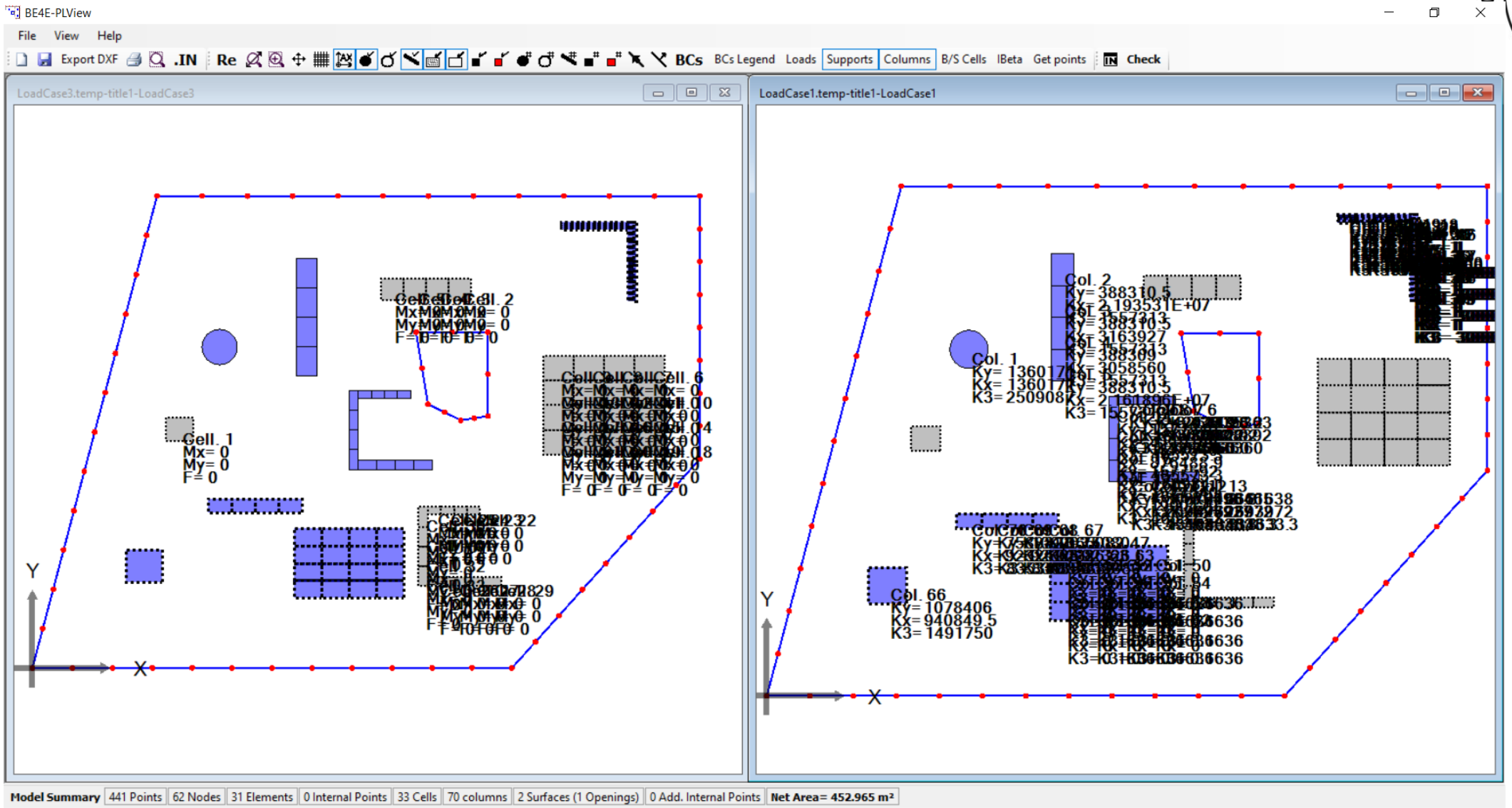


BE model showing BE number, directions, and normal directions

BE model showing boundary conditions

CUFE-BE

www.be4e.com

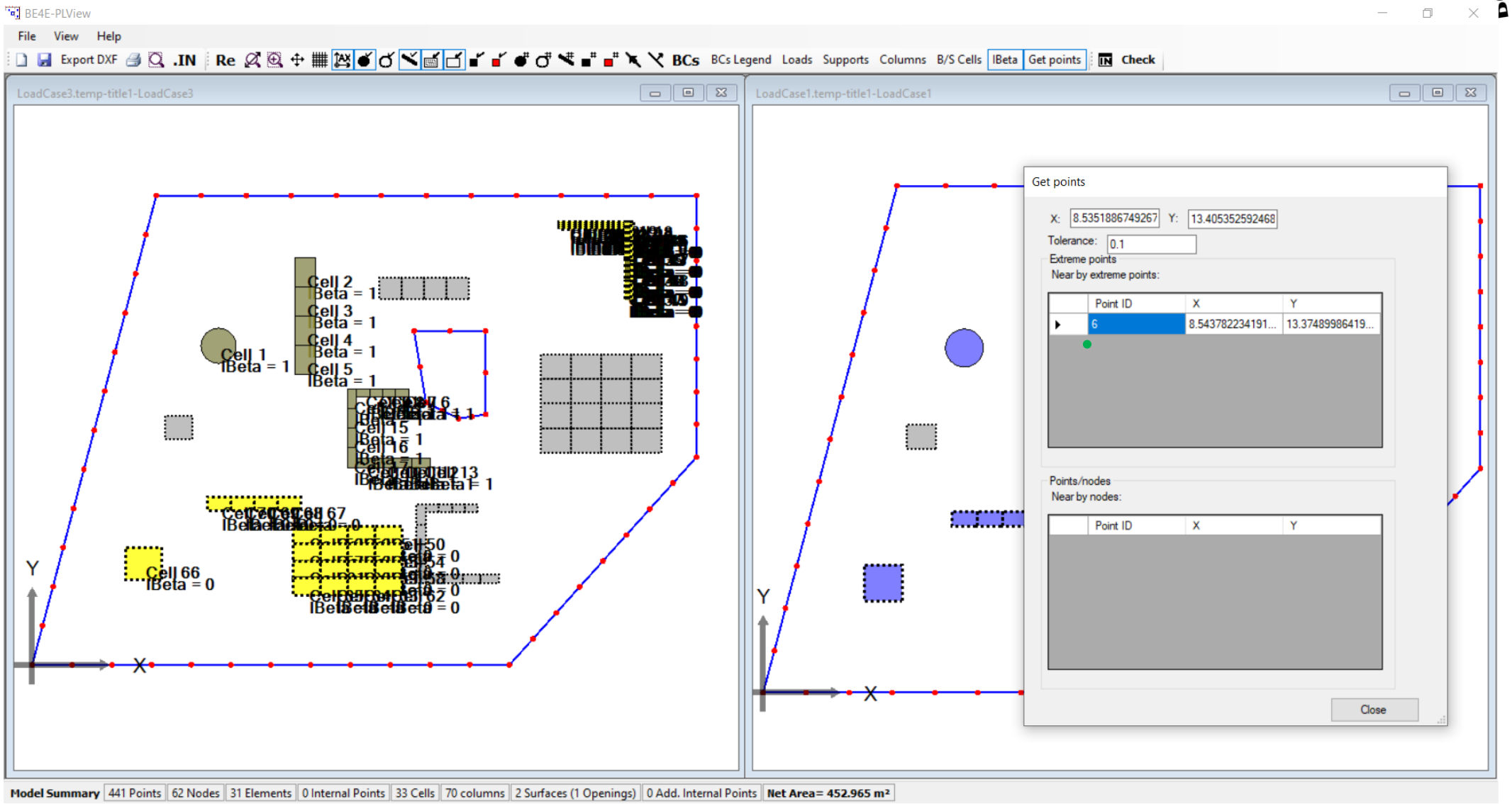


BE model showing load values on loading cells

BE model showing stiffness values on supporting cells

CUFE-BE

www.be4e.com

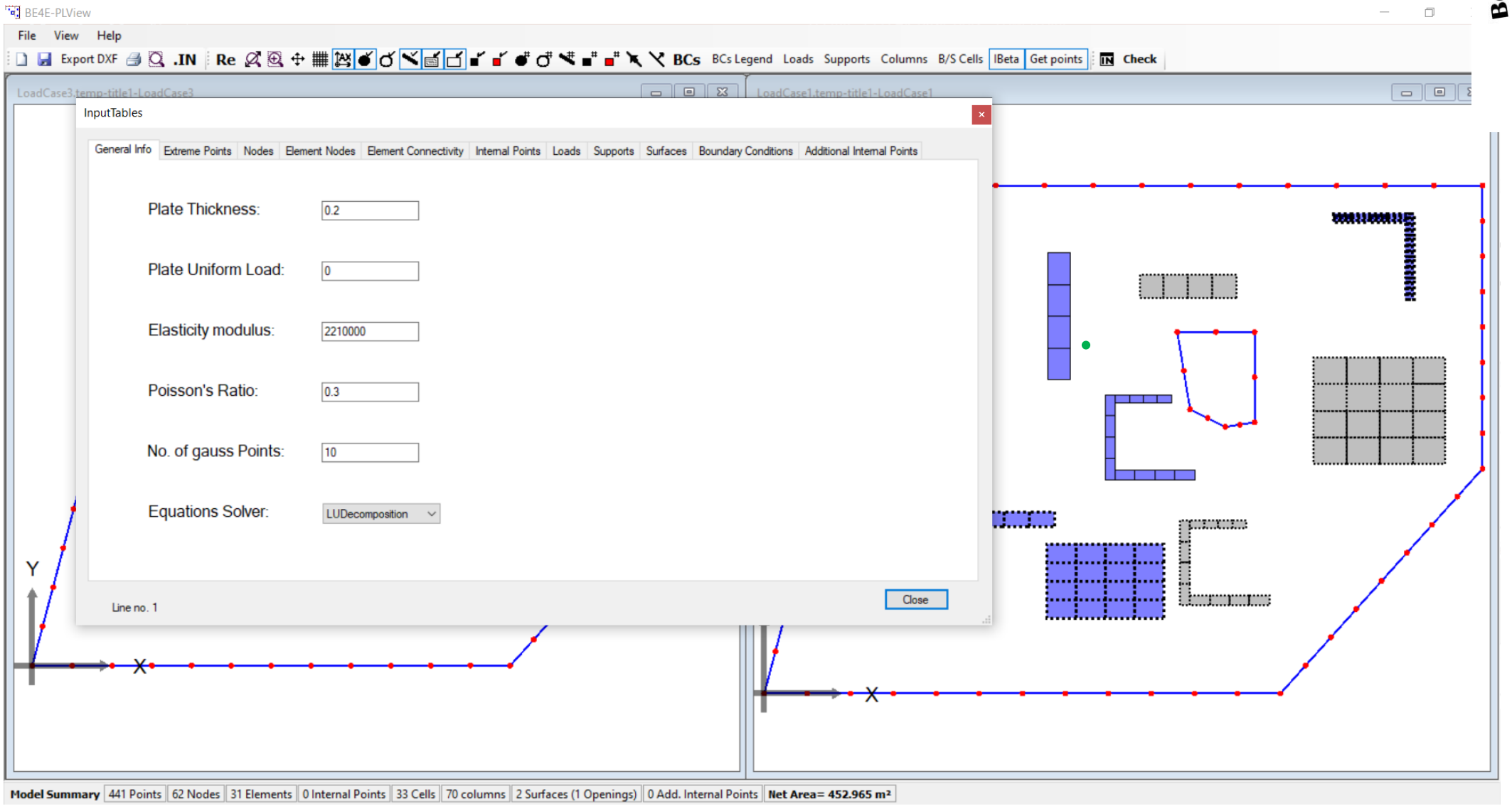


BE model showing IBeta values on supporting cells

BE model showing clicked extreme point coordinates

CUFE-BE

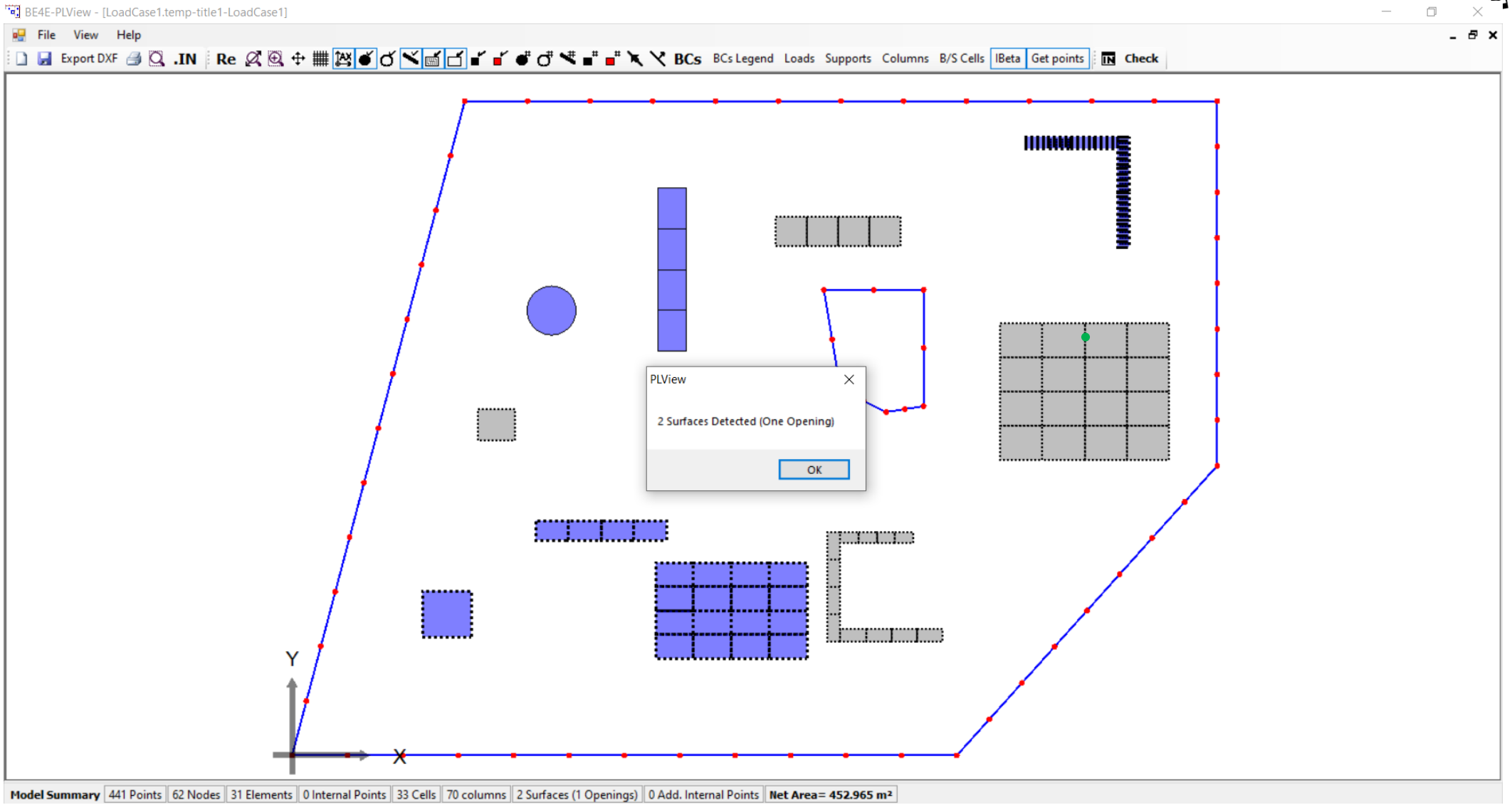
www.be4e.com



Tabulated values of the model

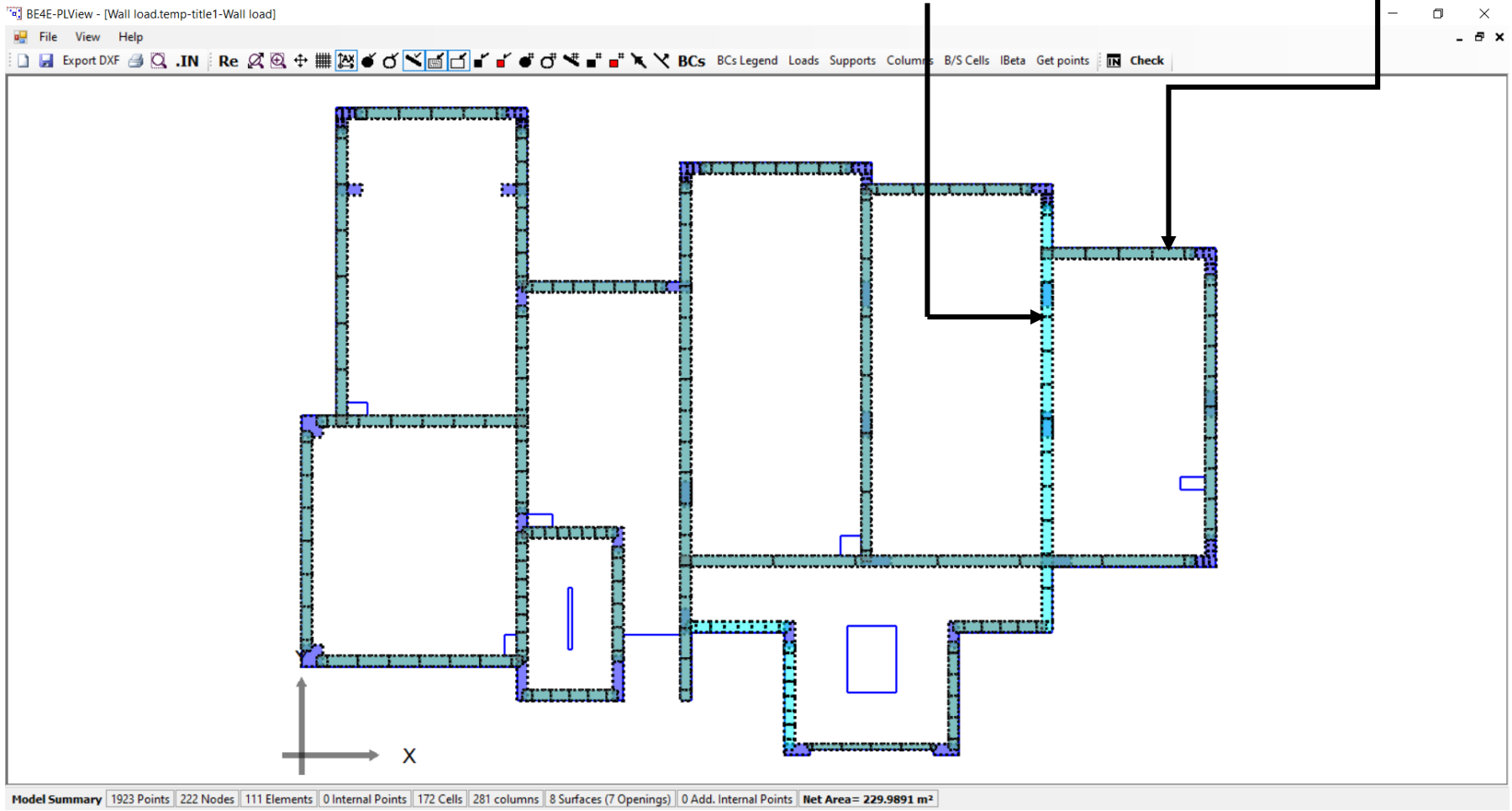
CUFE-BE

www.be4e.com



Number of surfaces in the model by clicking the Check button

- 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

Table of contents

CUFE-BE

www.be4e.com

Table of contents

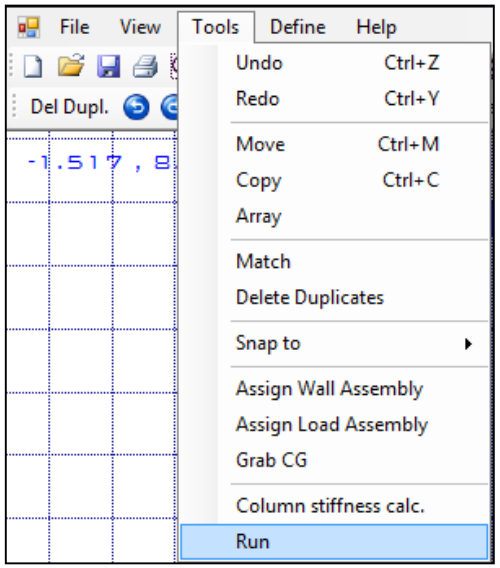
- PLCoreMan has several tasks:

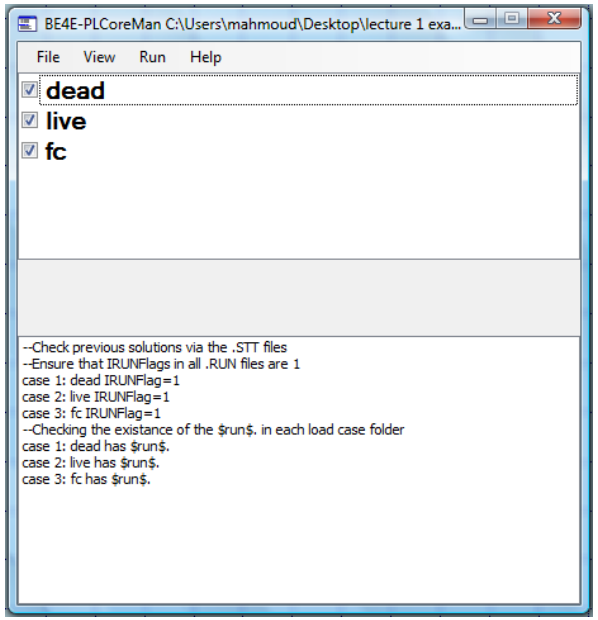
1- Run the model.

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



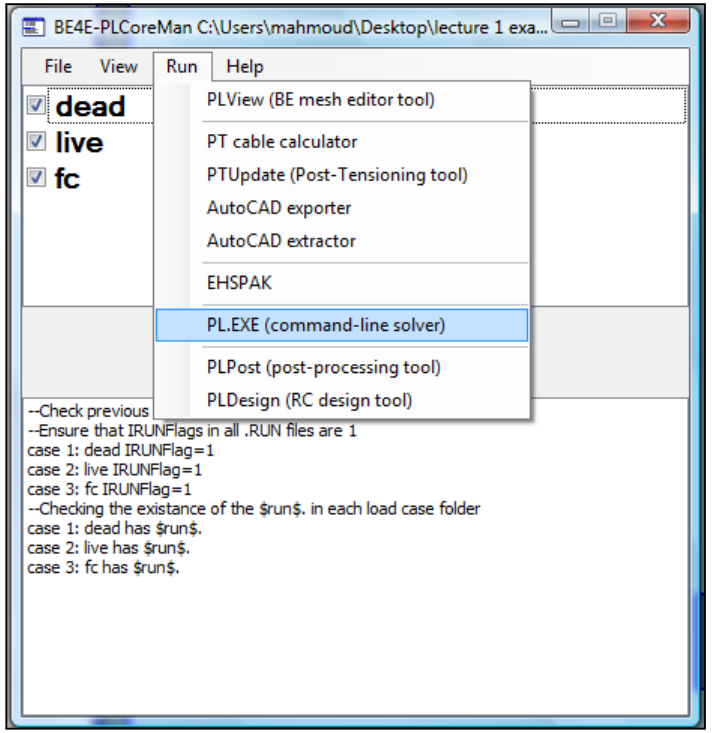


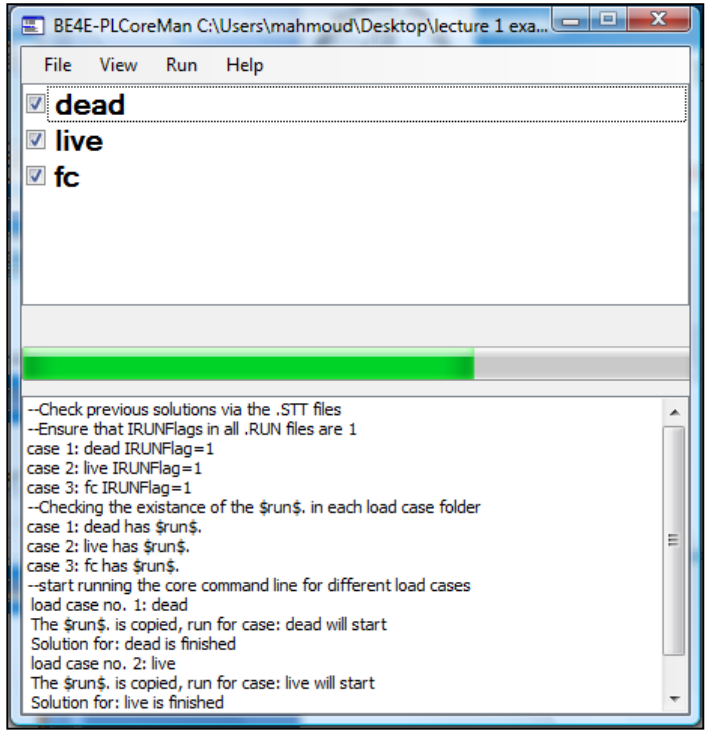
CUFE-BE

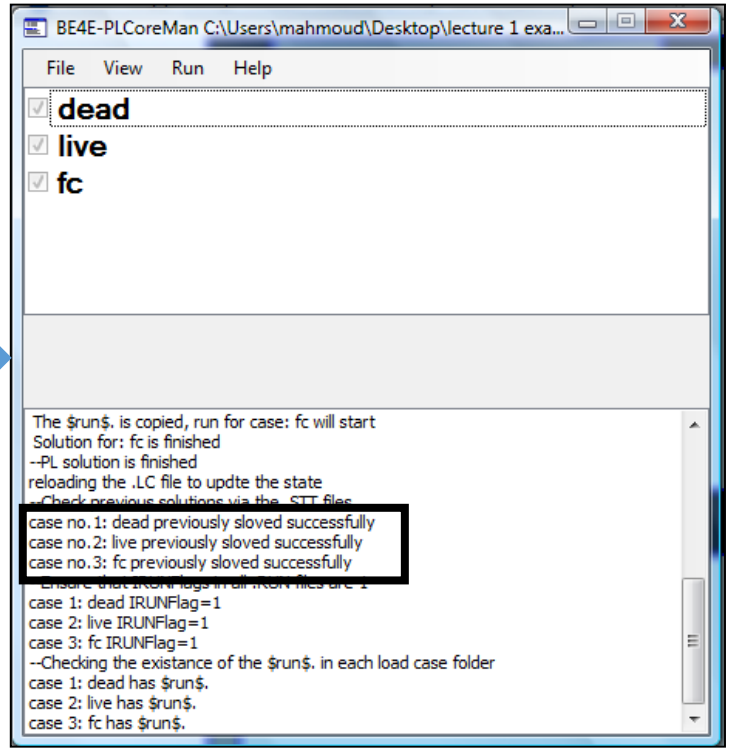
www.be4e.com

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.



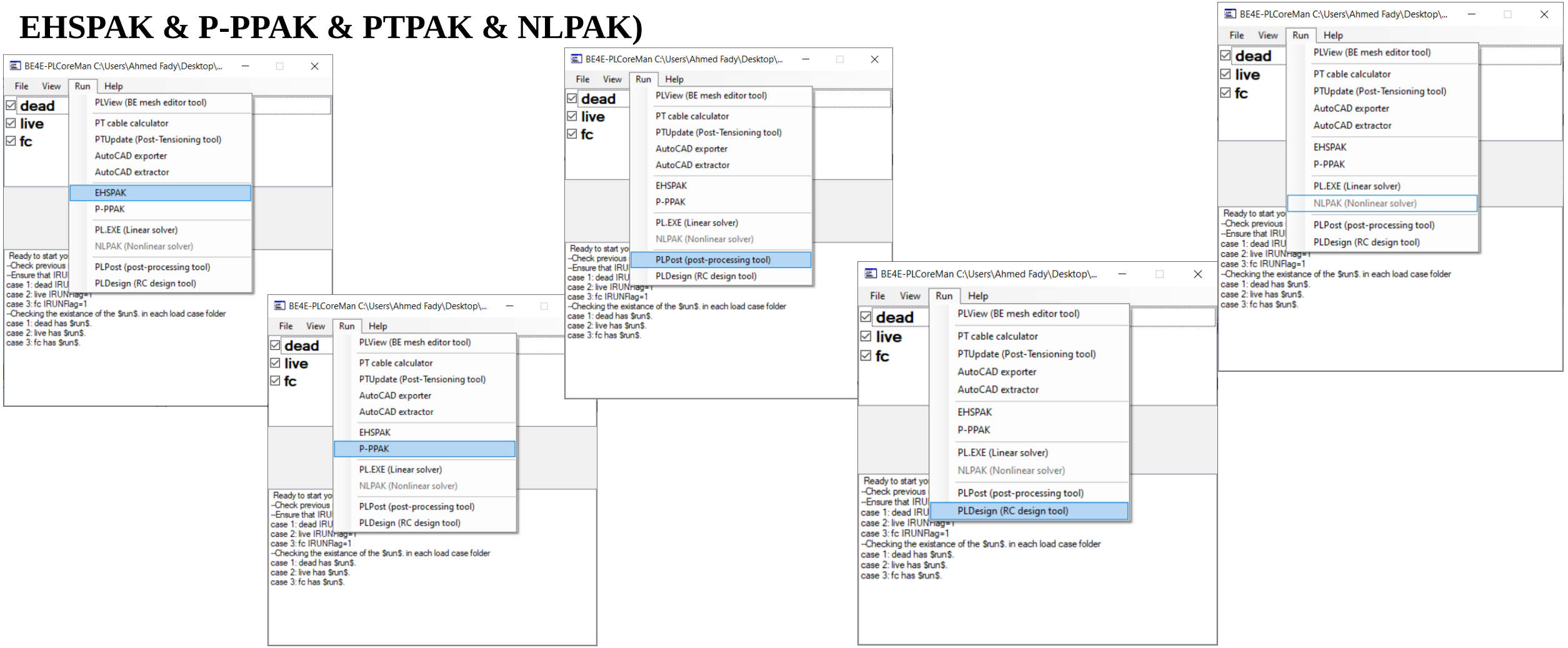




CUFE-BE

www.be4e.com

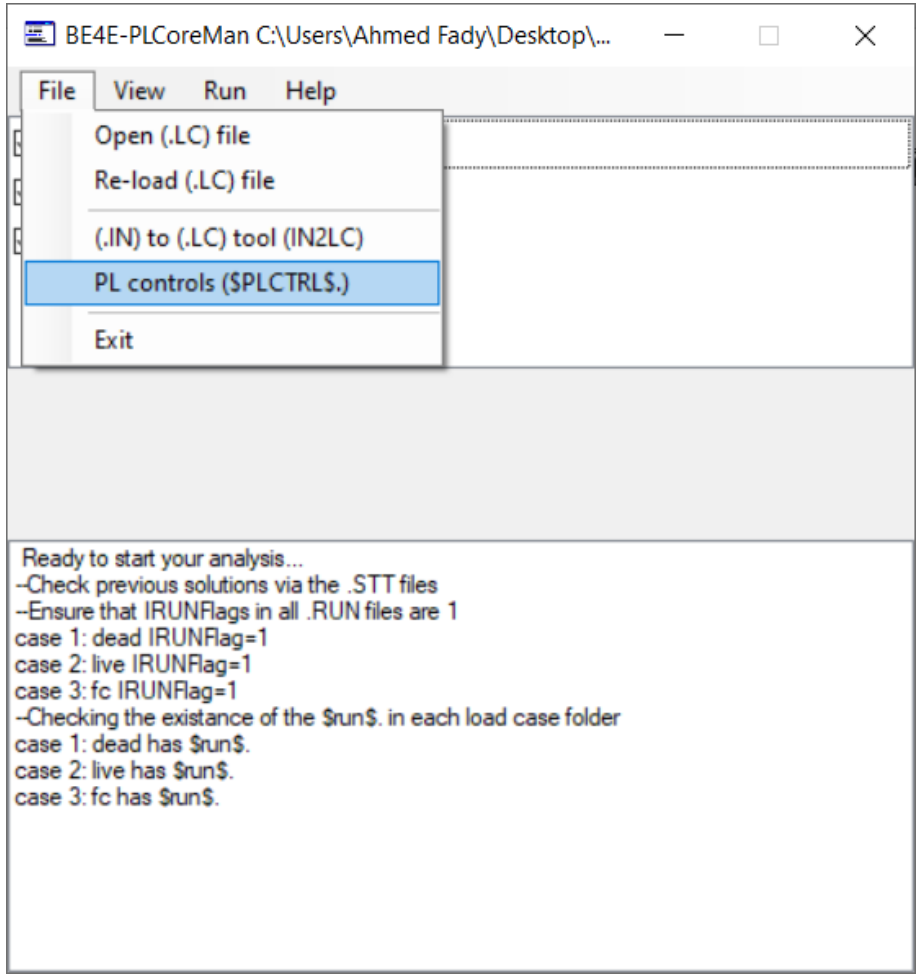
2- Transfer between PLPAK components (PLView & PLPost) or between other packages (PLDesign & EHSPAK & P-PPAK & PTPAK & NLPAK)

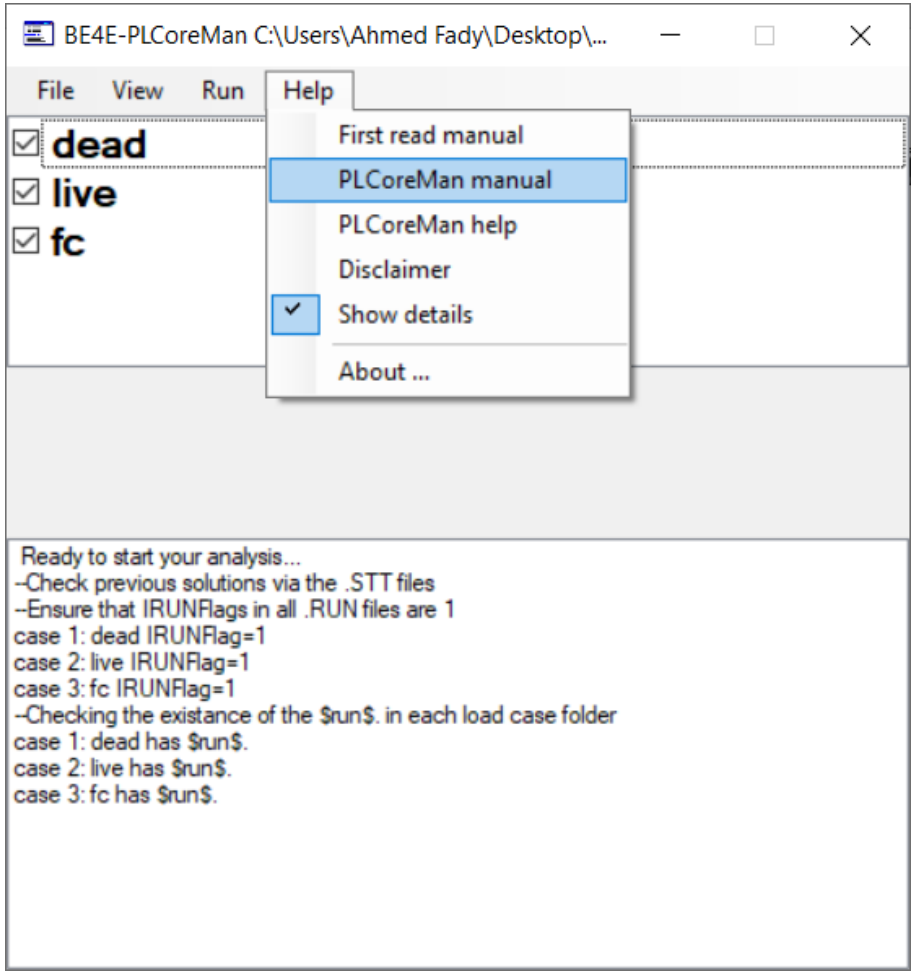


CUFE-BE

www.be4e.com

3- Open PL controls, manual, and about





CUFE-BE

www.be4e.com

Table of contents

CUFE-BE

www.be4e.com

Table of contents

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

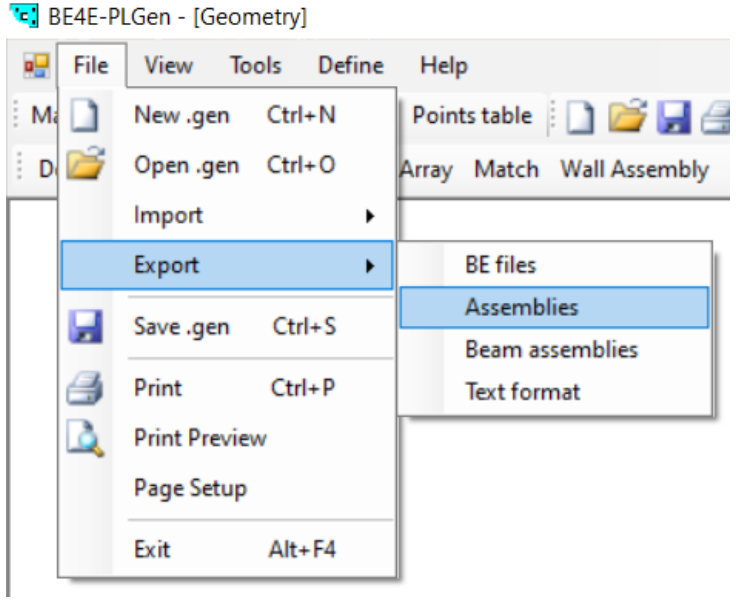
1. **Files exported from PLGen**
2. **Load combinations**
3. **Slab results**
    - **3.1. Contour results**
    - **3.2. Query result**
    - **3.3. Strip results**
    - **3.4. Supporting elements results**
4. **Beam results**

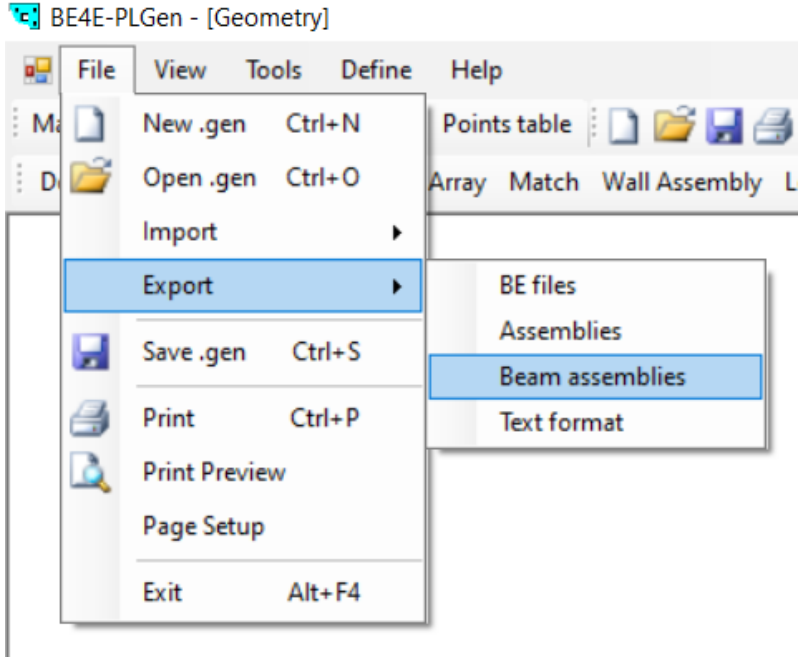
1. Files exported from PLGen

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.

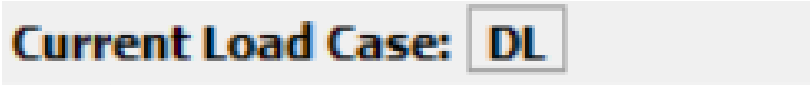
Restore these files in the PLPost will be demonstrated later.





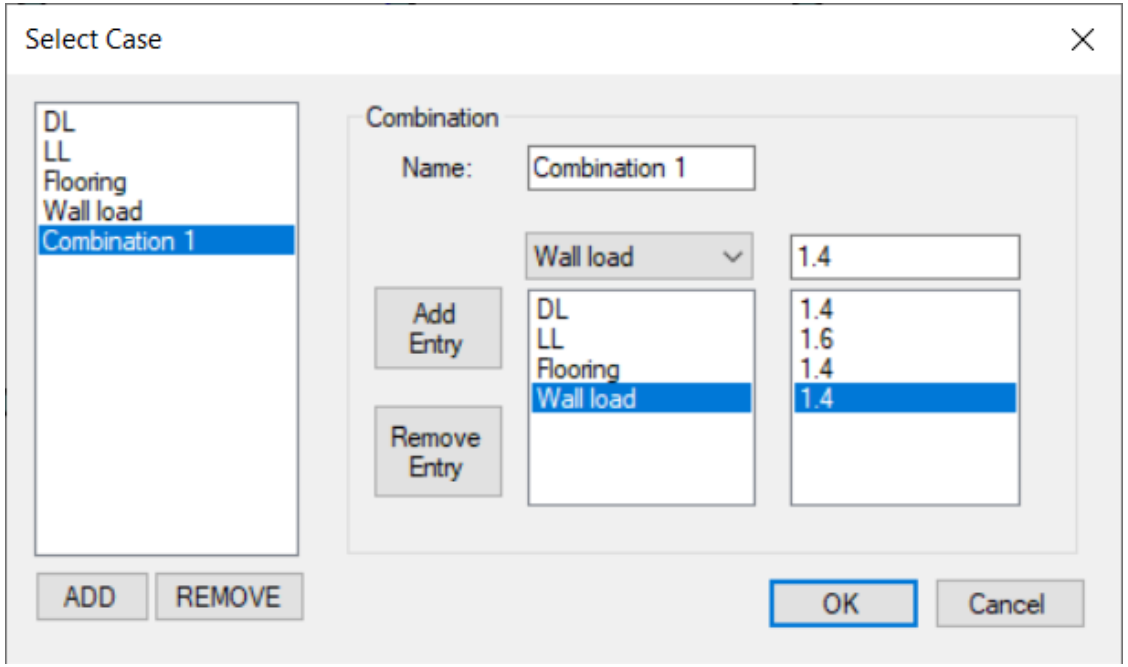
2. Load combinations

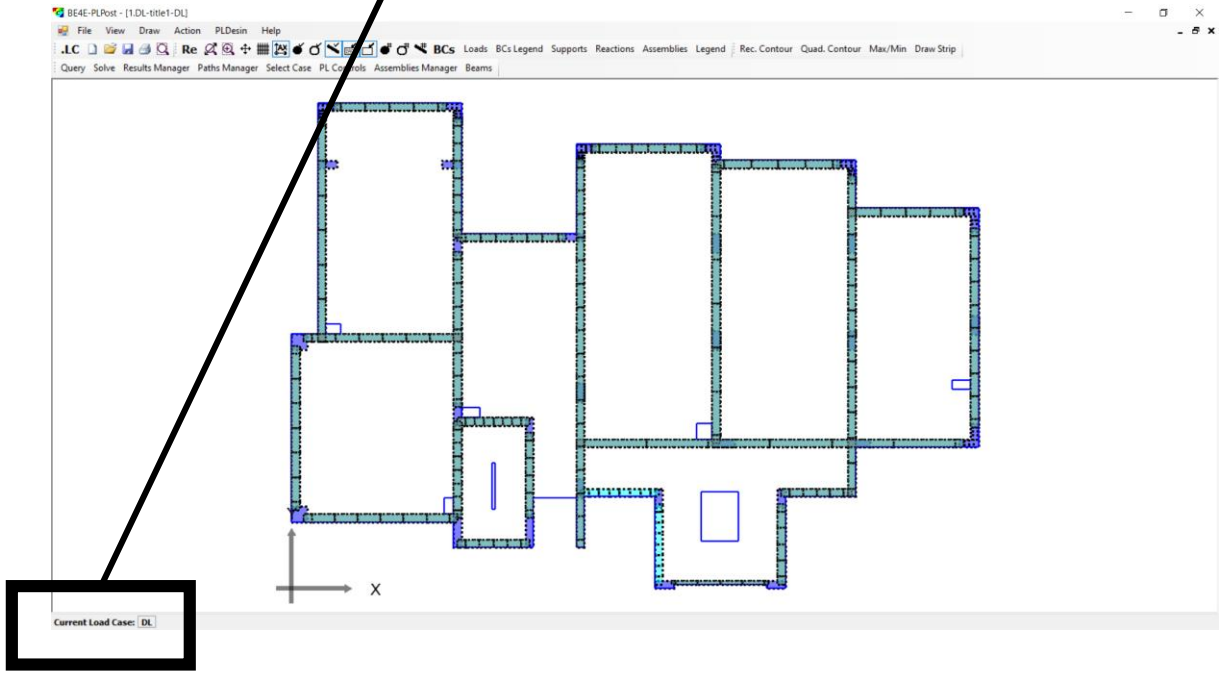
The lower tabs of the PLPost contain by default current load case (by default it is the first load case), can be changed by double click on it.



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

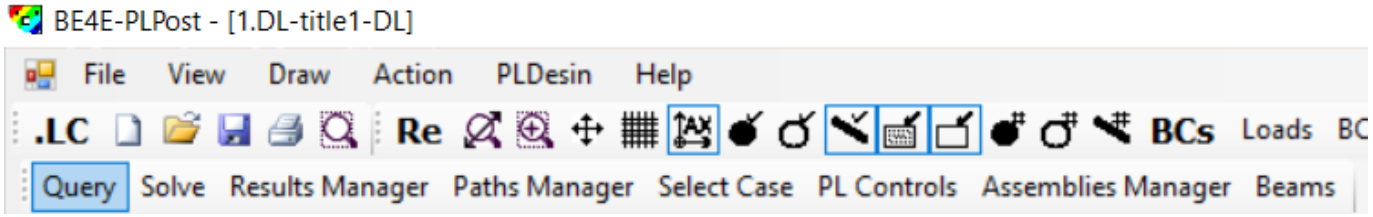




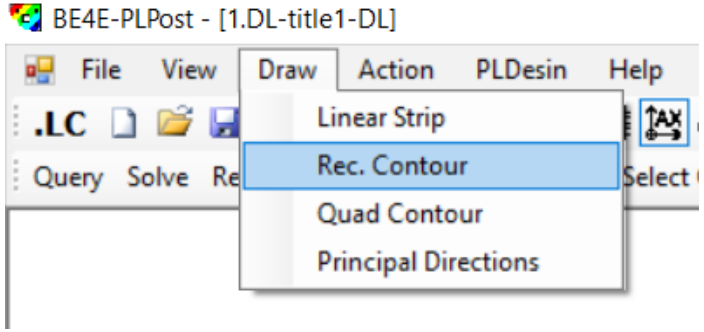
3. Slab results

There are 3 types for showing results in slab:

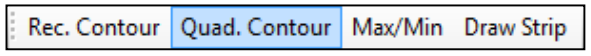
- Contour (main, quadratic, rectangular)
- Strip
- Query for specific point

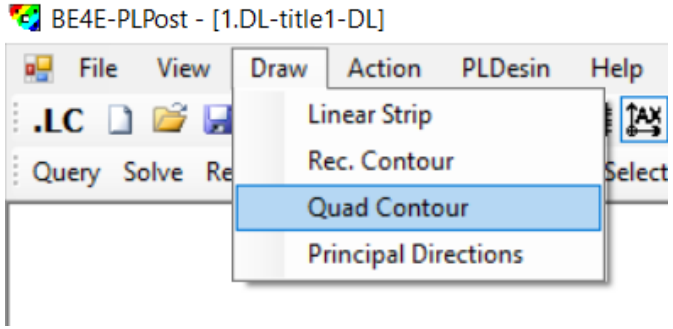


Draw rectangular contour

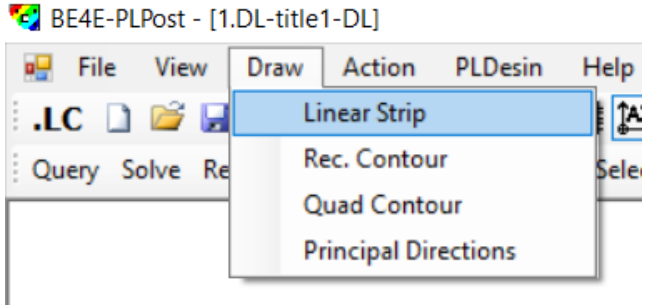


Draw quadratic contour





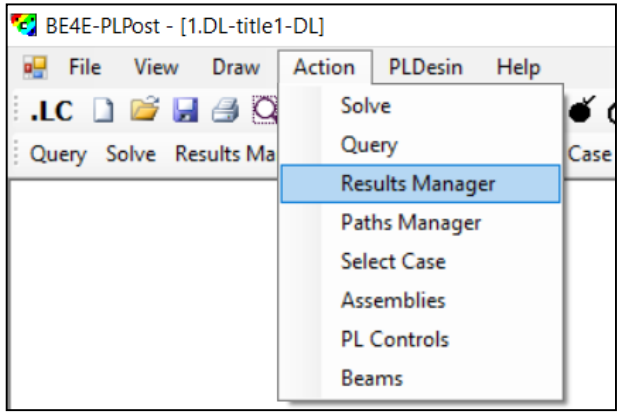
Draw strip



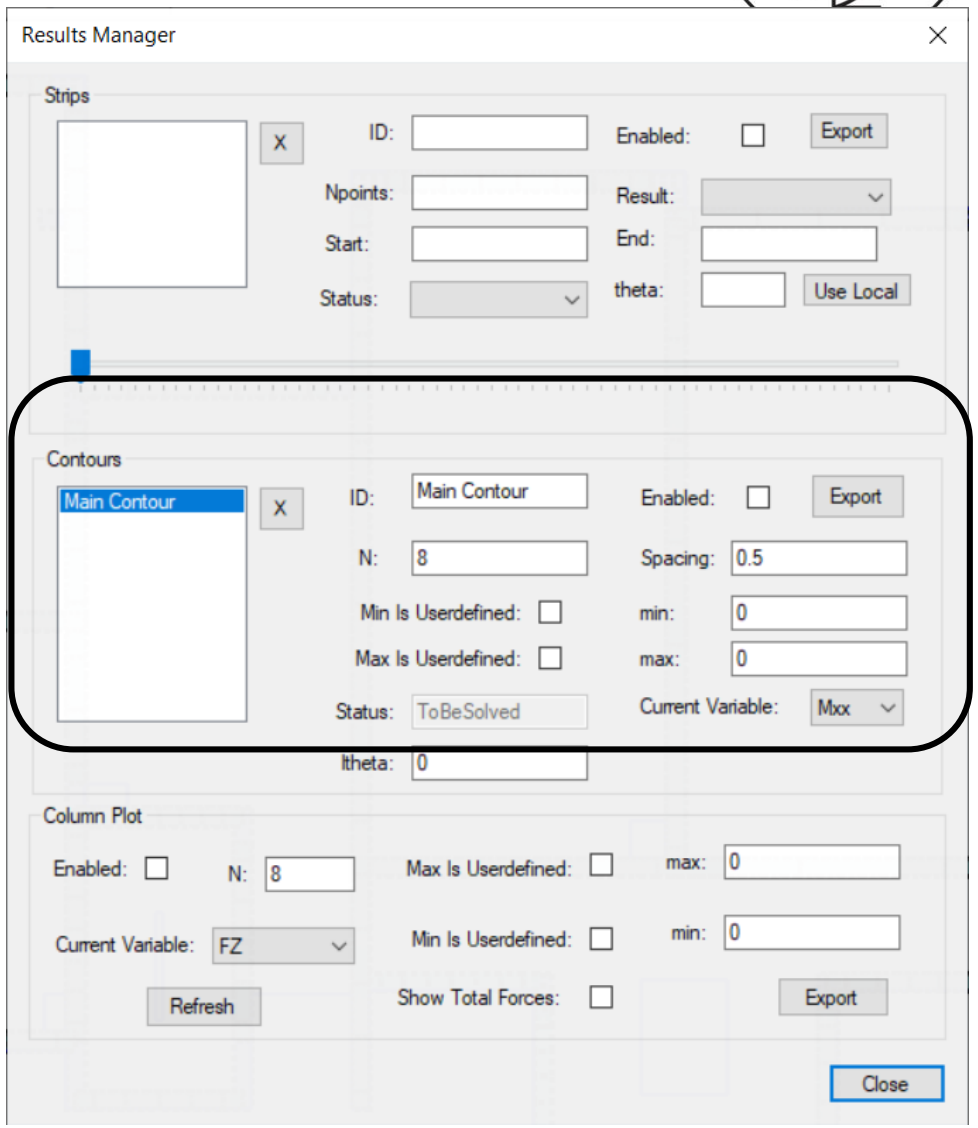
3. Slab results – contour results

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

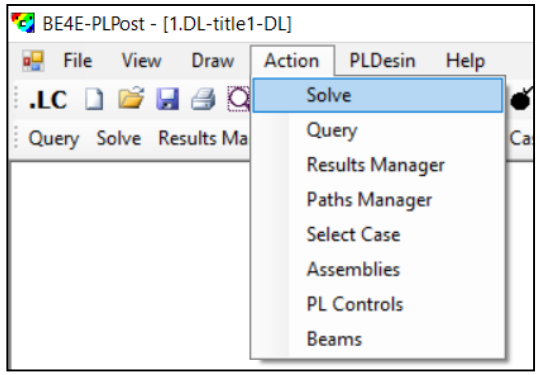


CUFE-BE

www.be4e.com

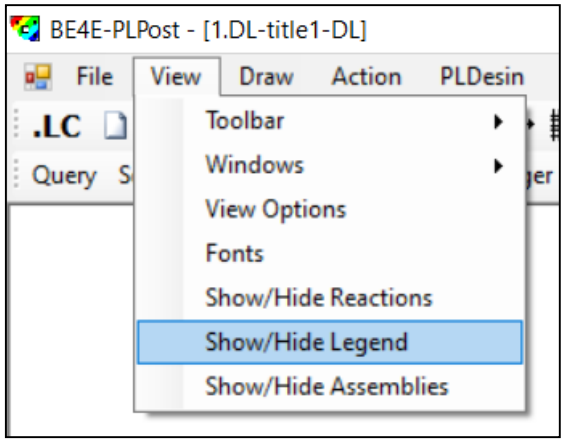
3. Slab results – contour results

Straining action for main contour using result manager:

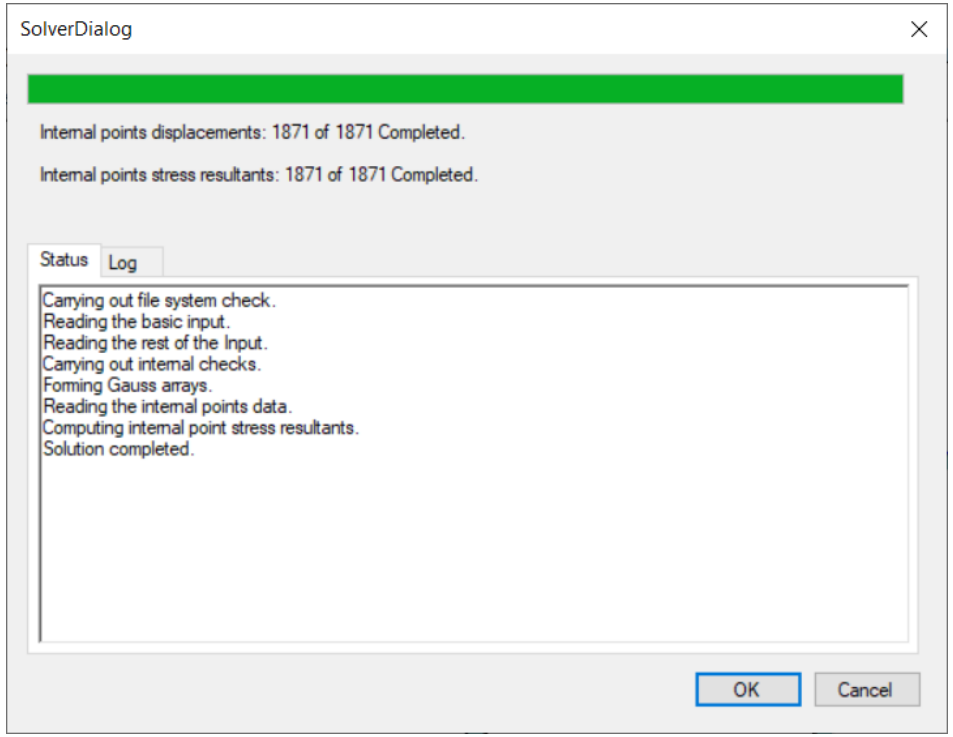


OR

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



OR



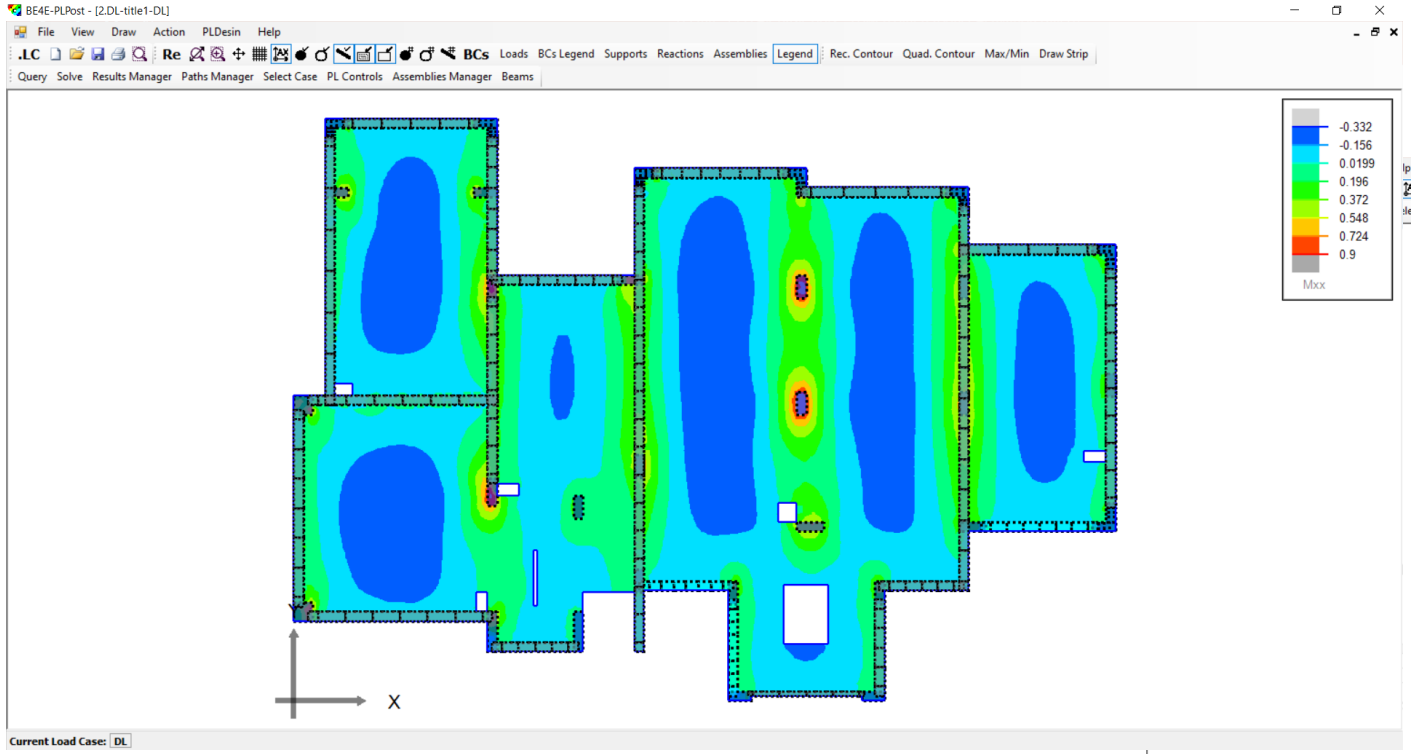
CUFE-BE

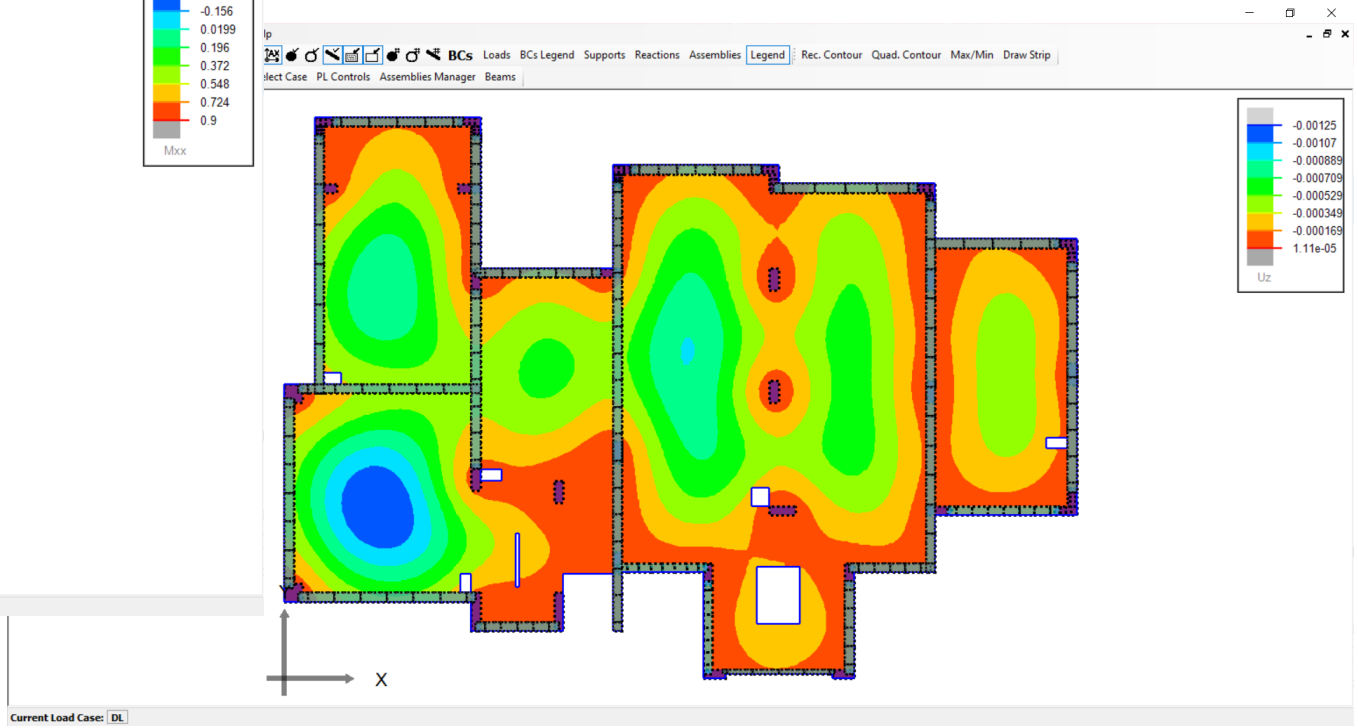
www.be4e.com

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.

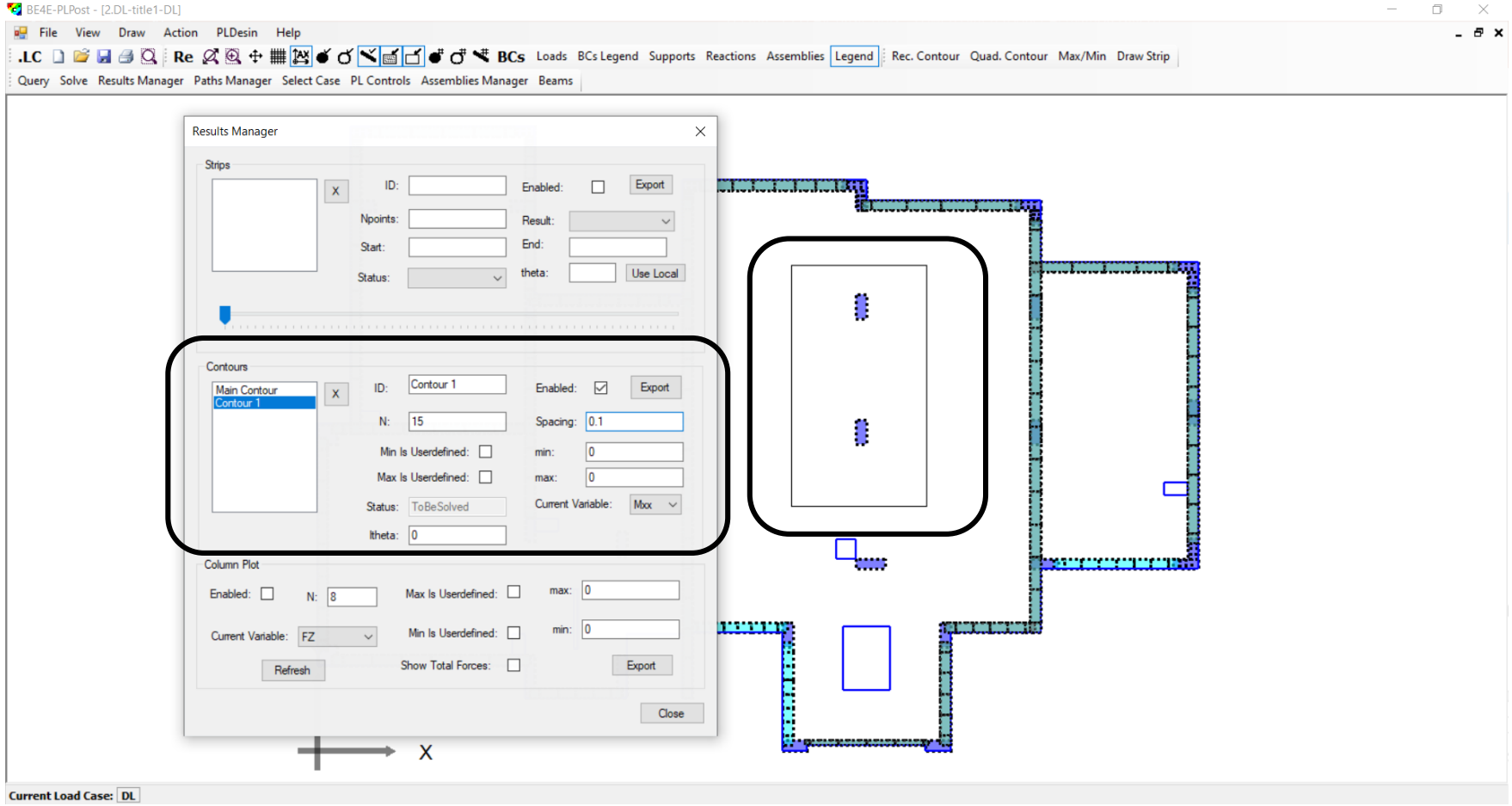




3. Slab results – contour results

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.



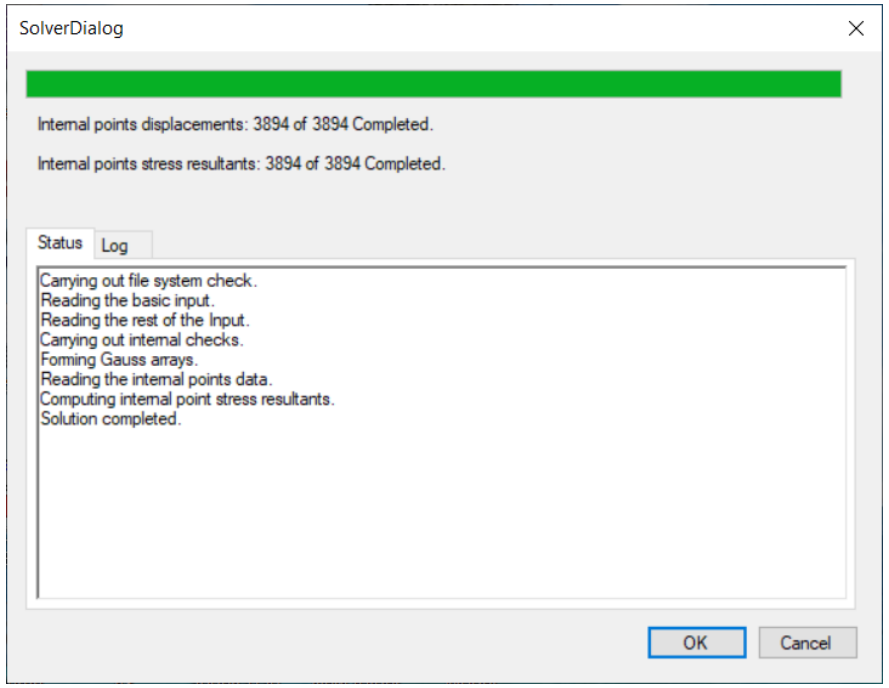
CUFE-BE

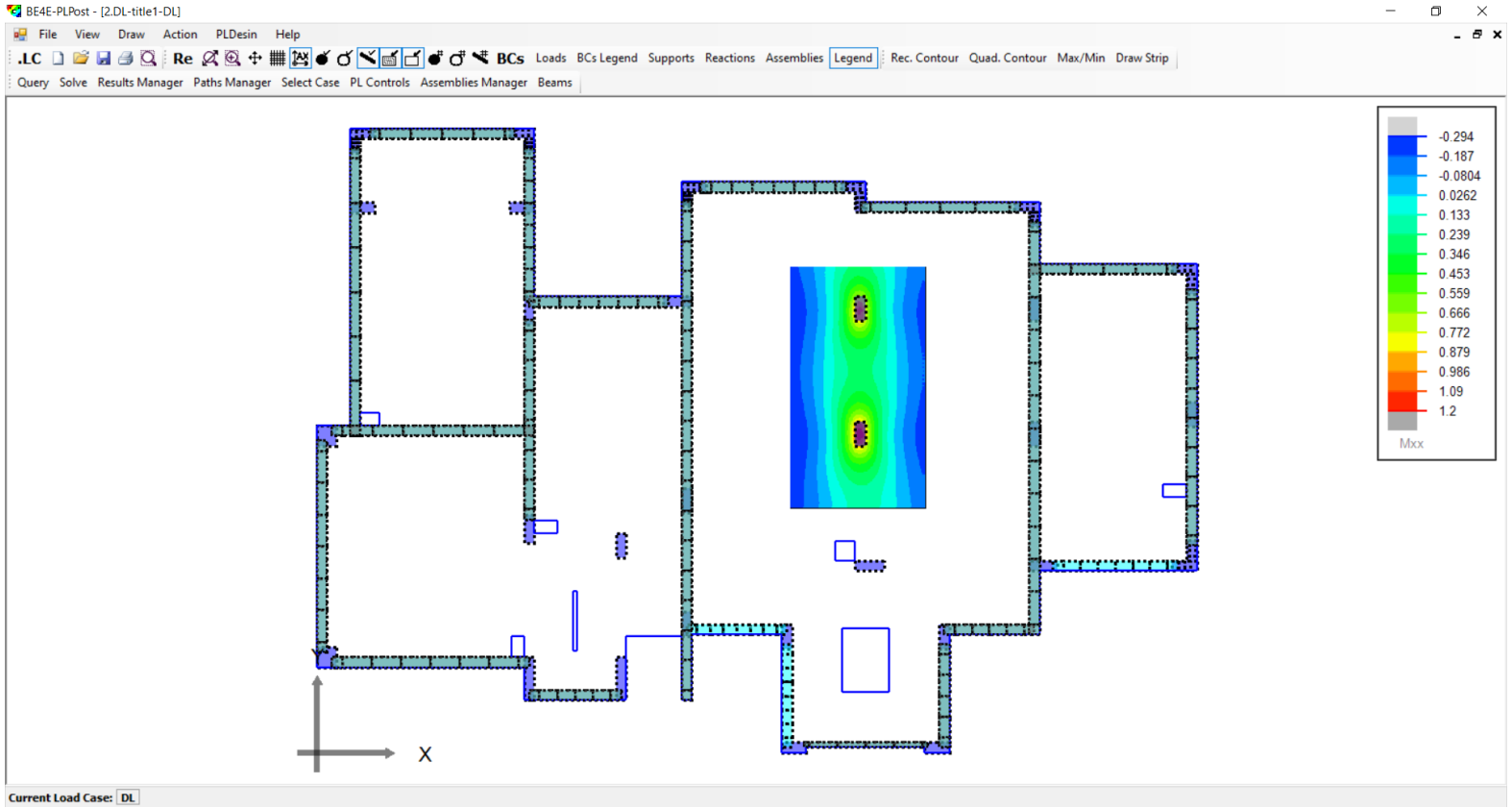
www.be4e.com

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.

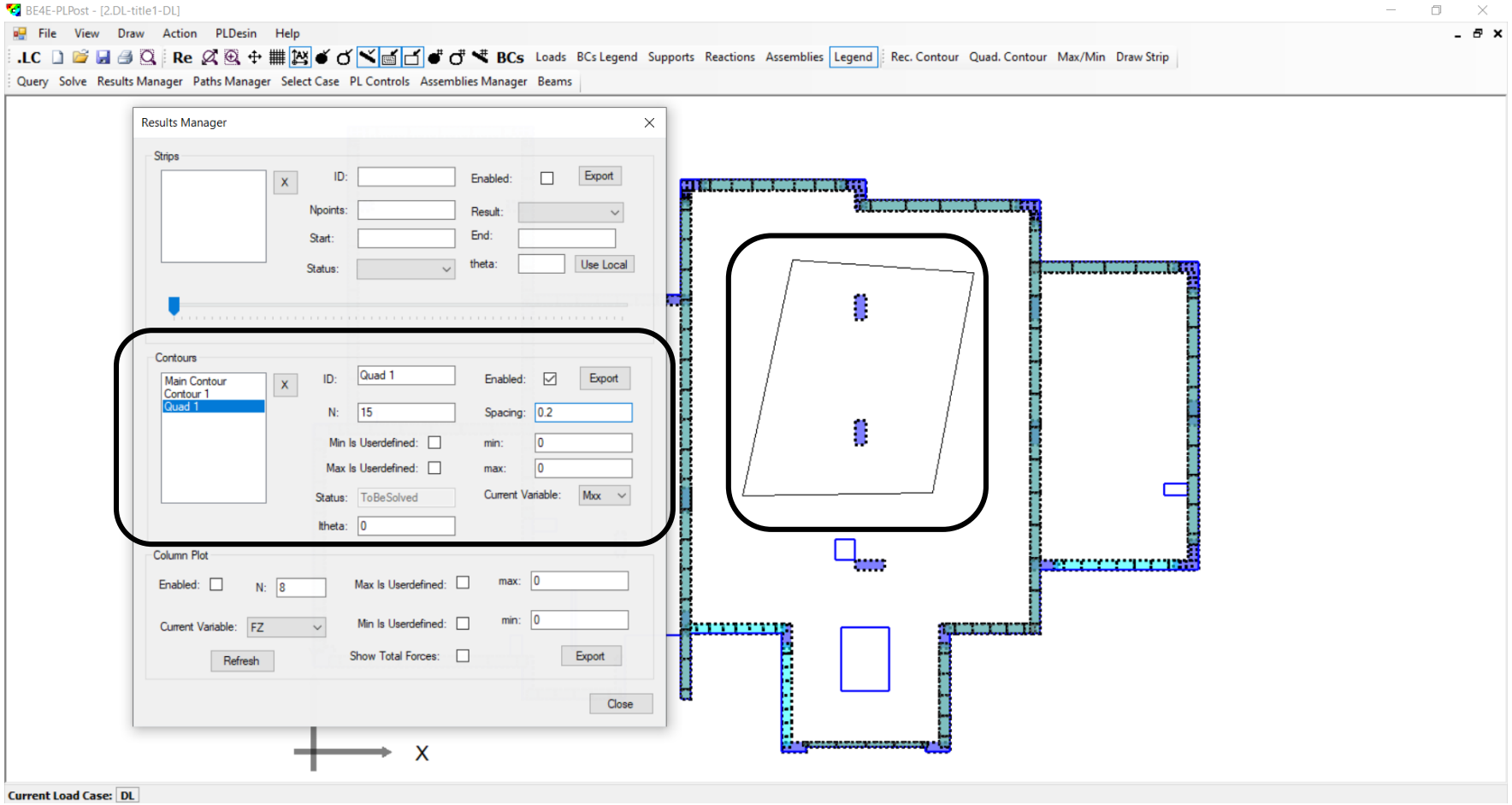




3. Slab results – contour results

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.



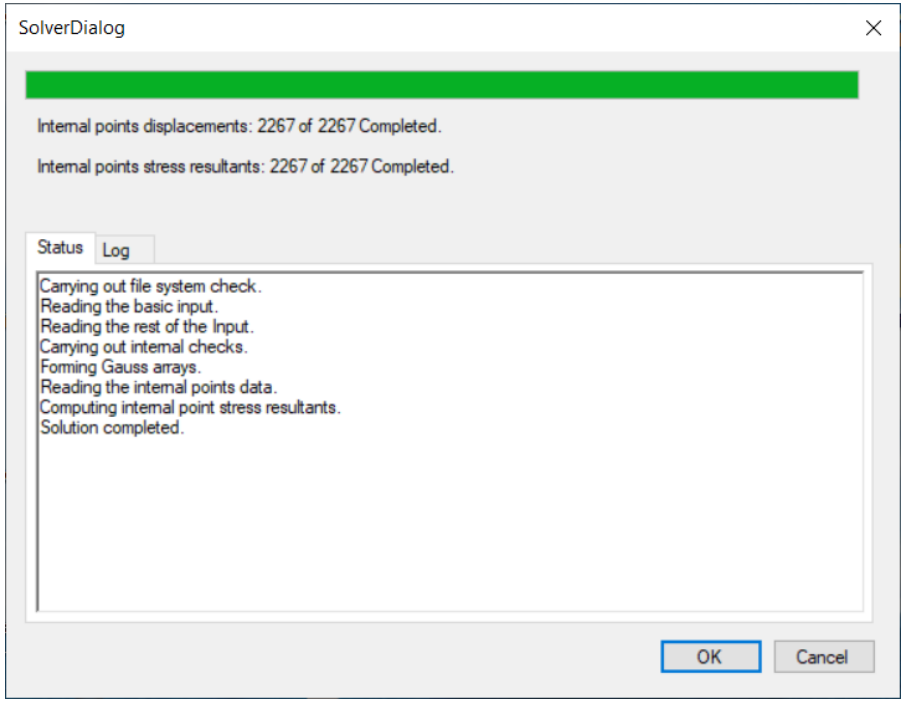
CUFE-BE

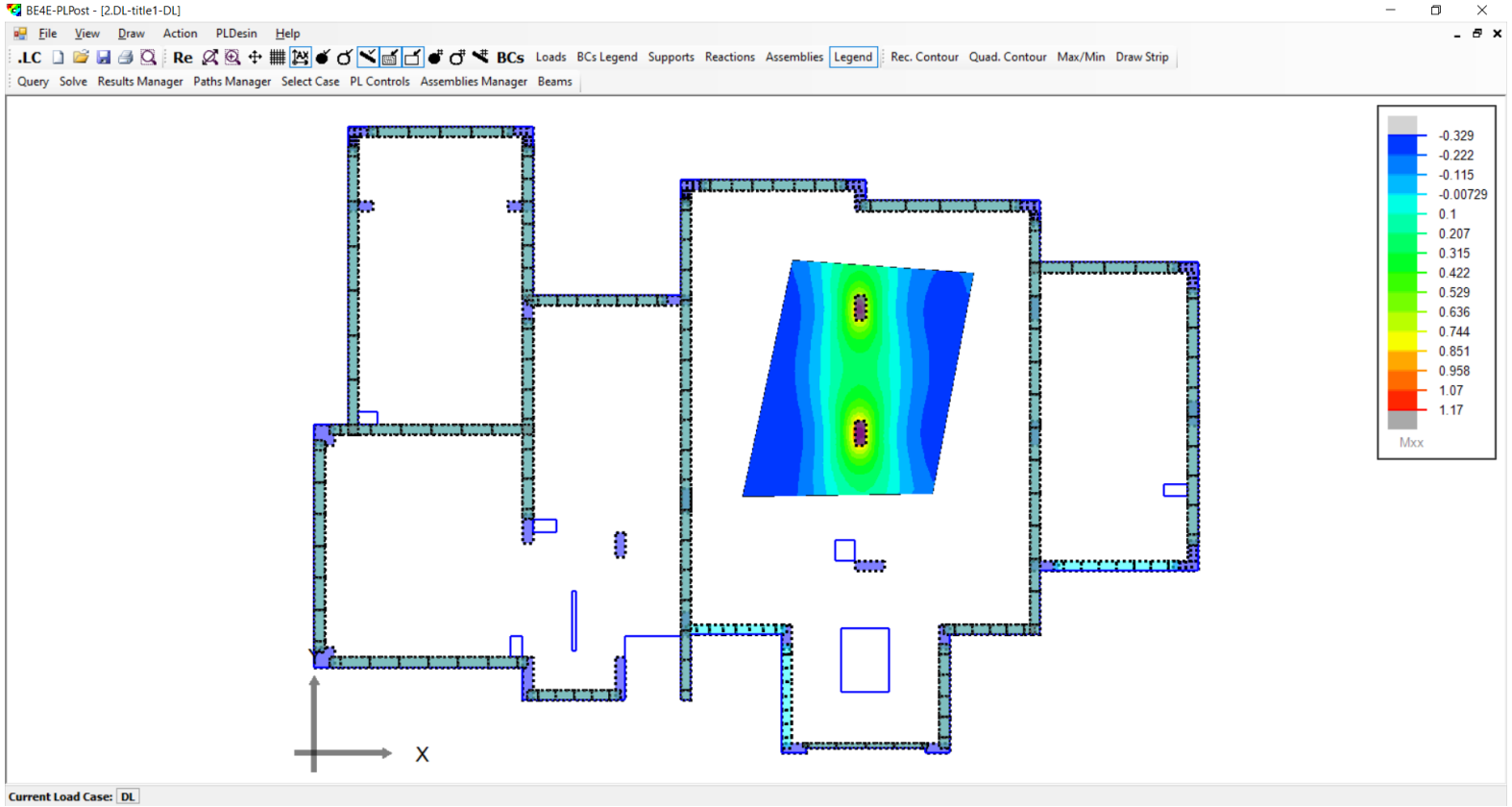
www.be4e.com

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.

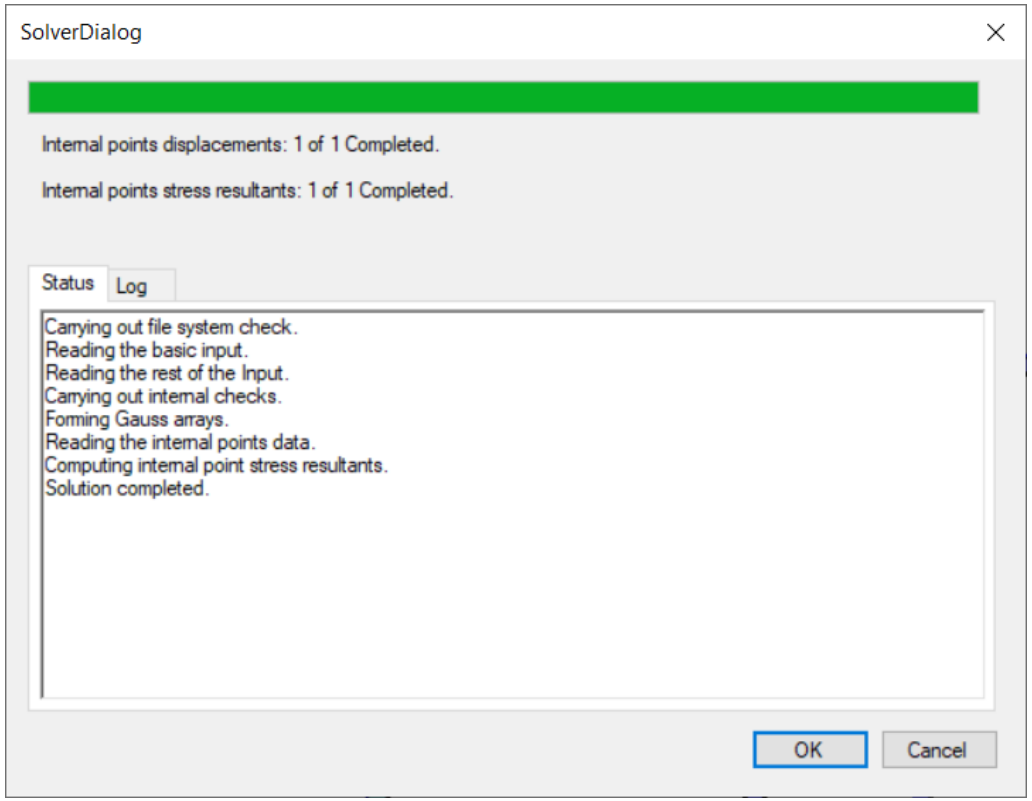


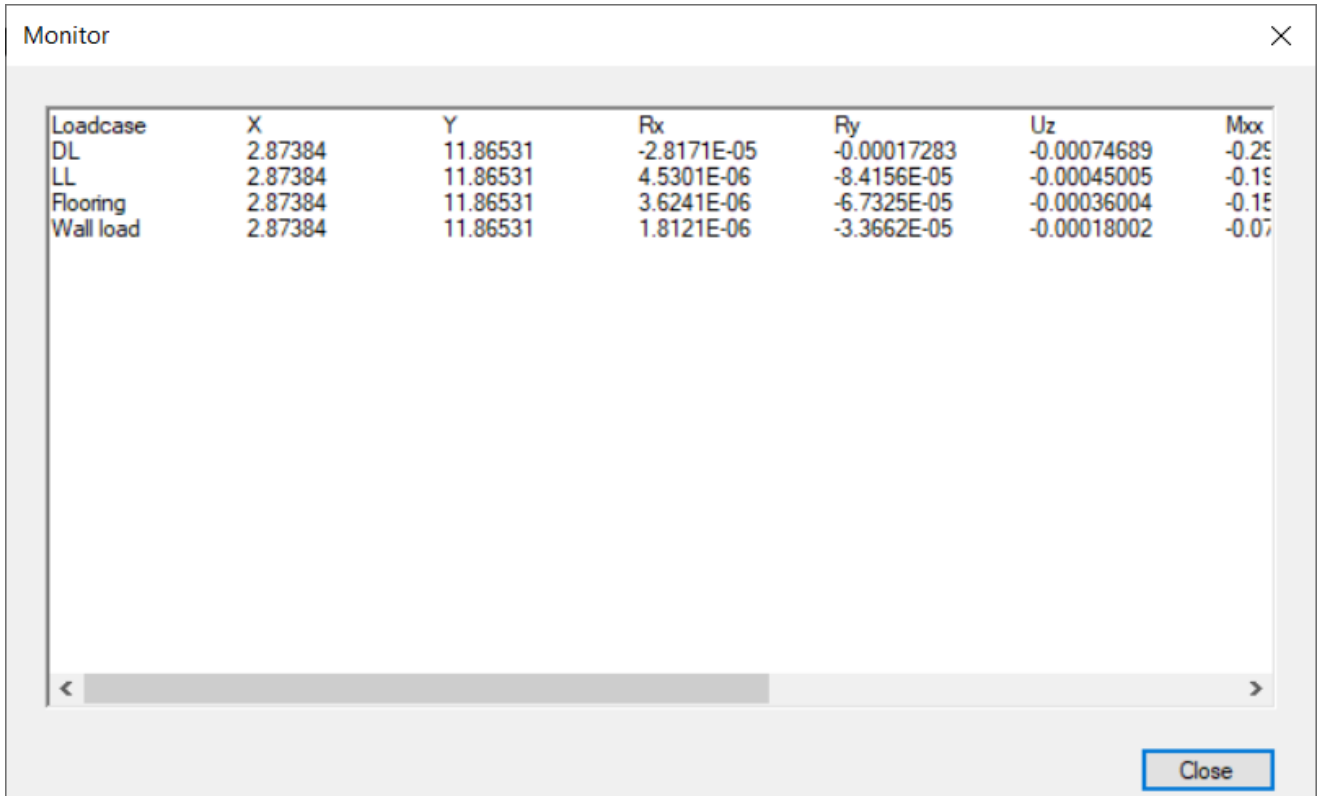


3. Slab results – query result

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.





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

CUFE-BE

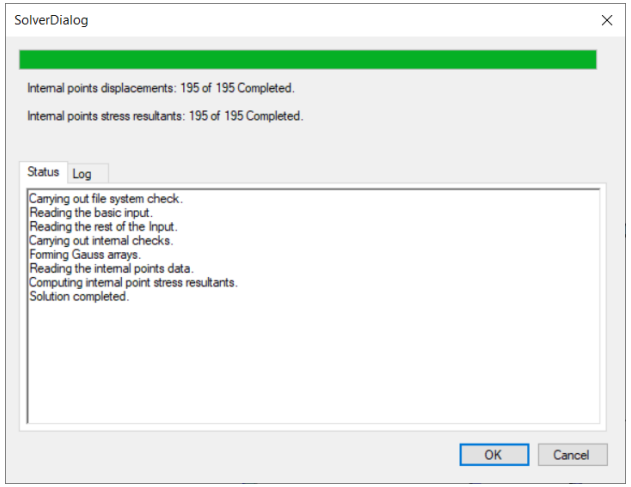
www.be4e.com

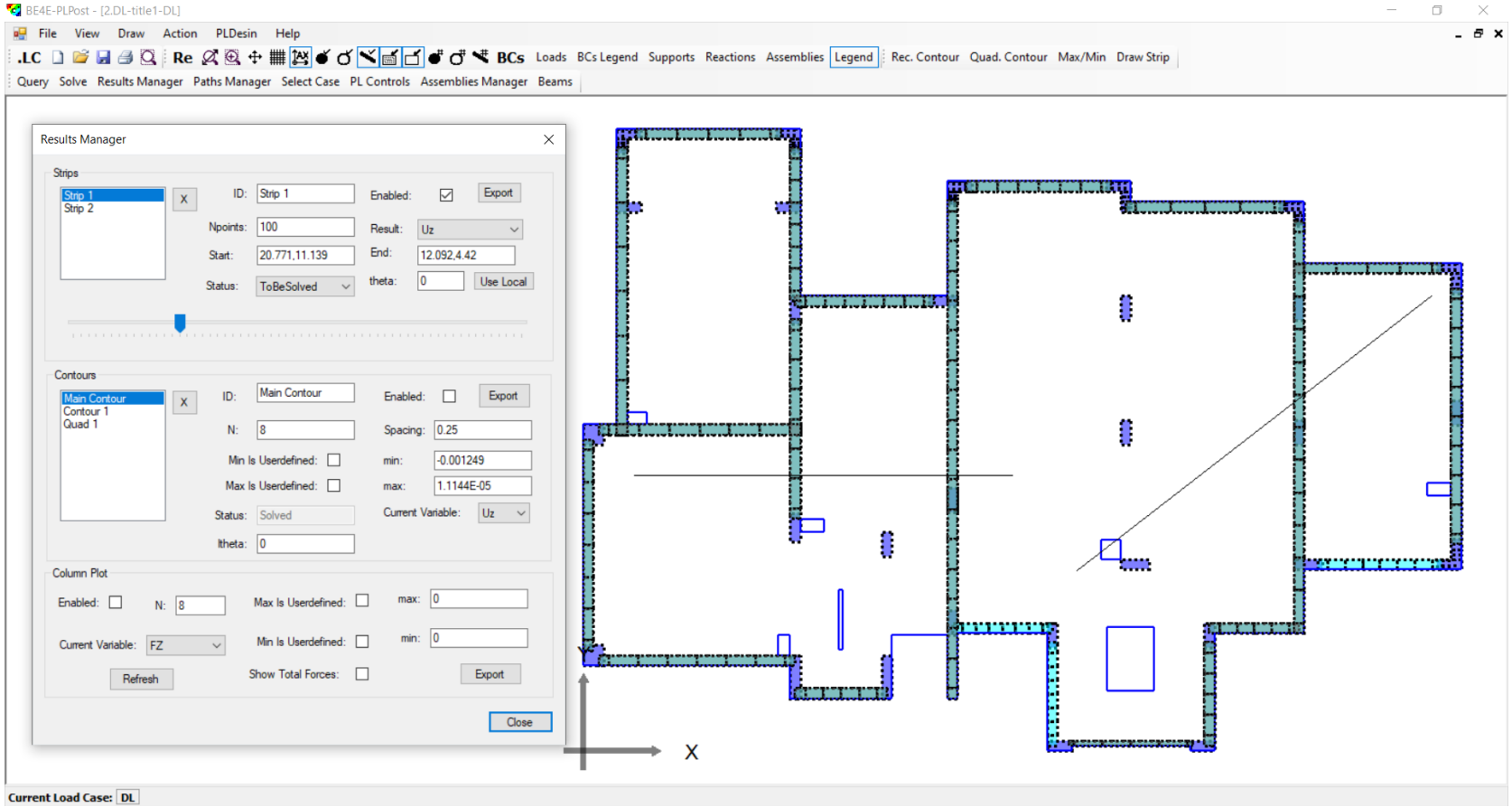
3. Slab results - strip result

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.



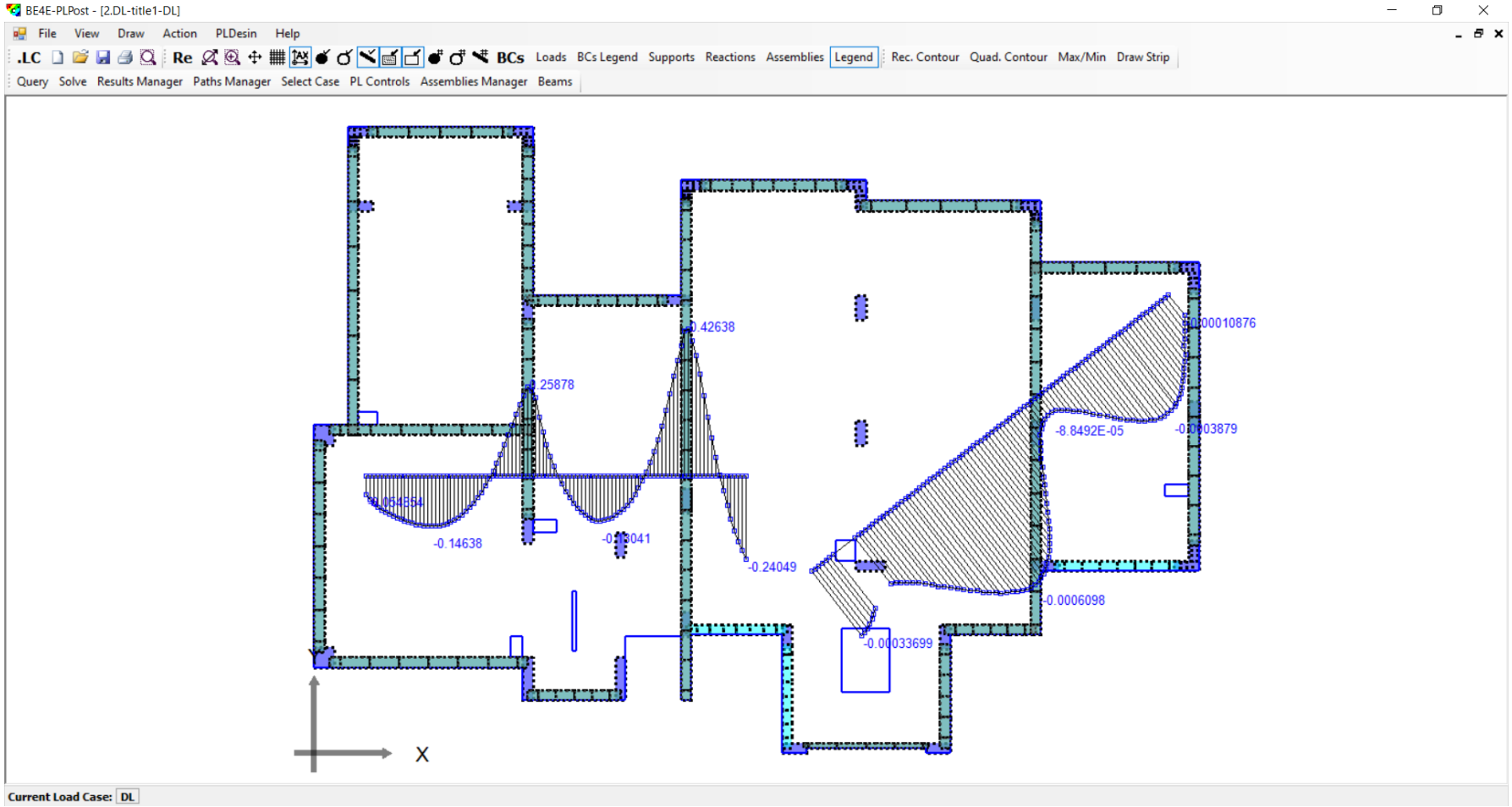


CUFE-BE

www.be4e.com

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.



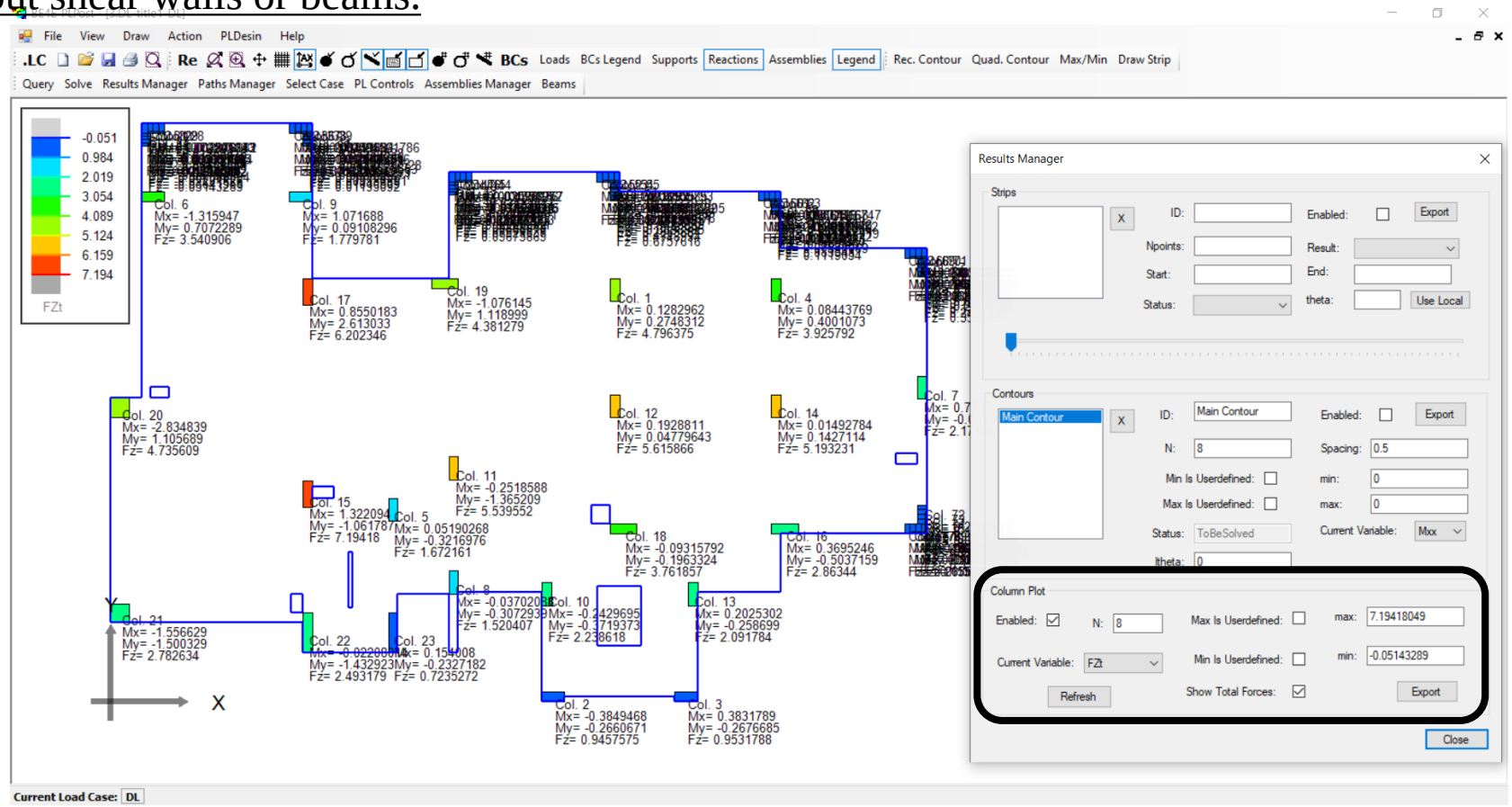
CUFE-BE

www.be4e.com

3. Slab results - supporting elements results

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

- <u>In case of quadratic columns and without shear walls or beams.</u>



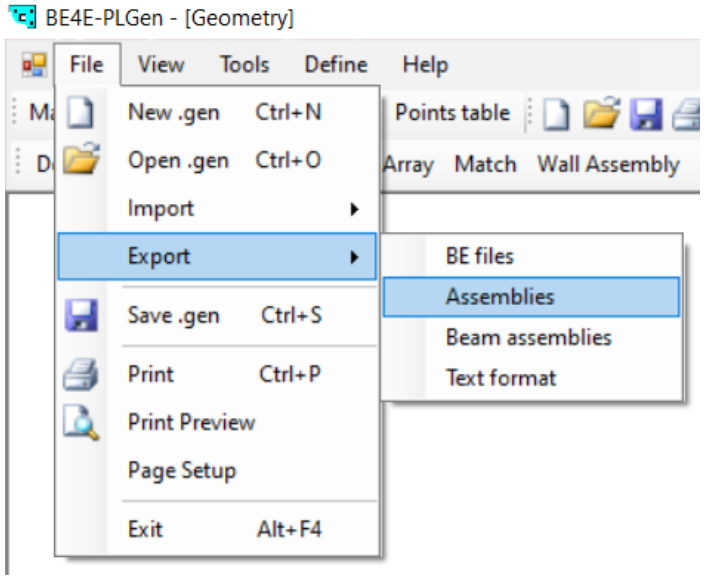
- Open the Result Manager, the last part is for column plot.
- The user mark on the check box (Enable).
- Mark on the check box (Show Total Force).
- Choose the Current Variable.
- The user can export the column straining
- actions on text file.
- Unlike slabs, the columns don't have to run.

3. Slab results - supporting elements results

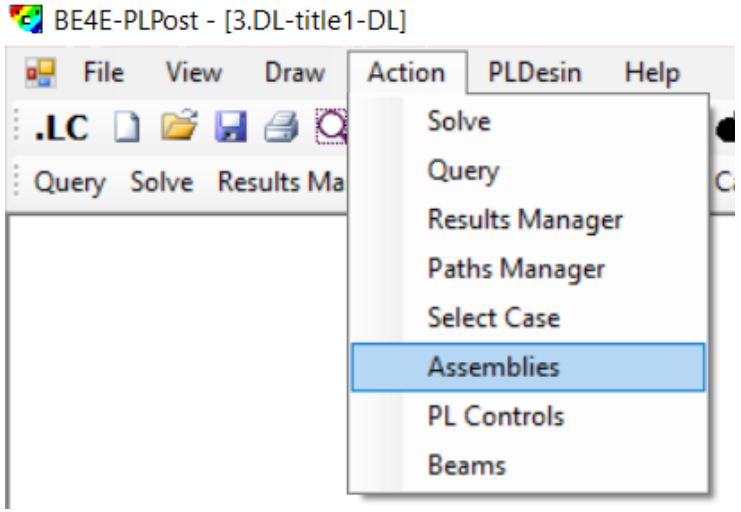
The user can show column analysis by two ways according to the type of model:

- <u>In case of columns more than four sides or slab with shear walls or beams.</u>

- 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



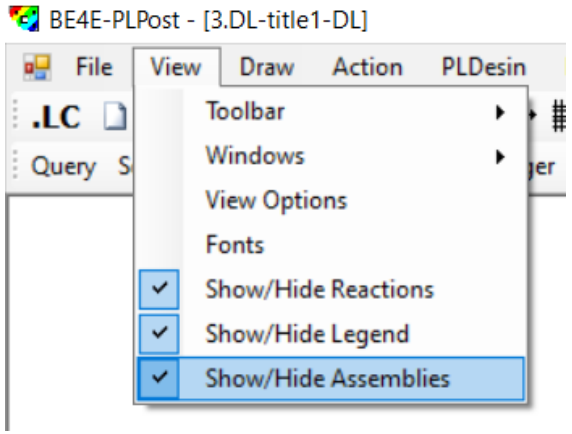
CUFE-BE

www.be4e.com

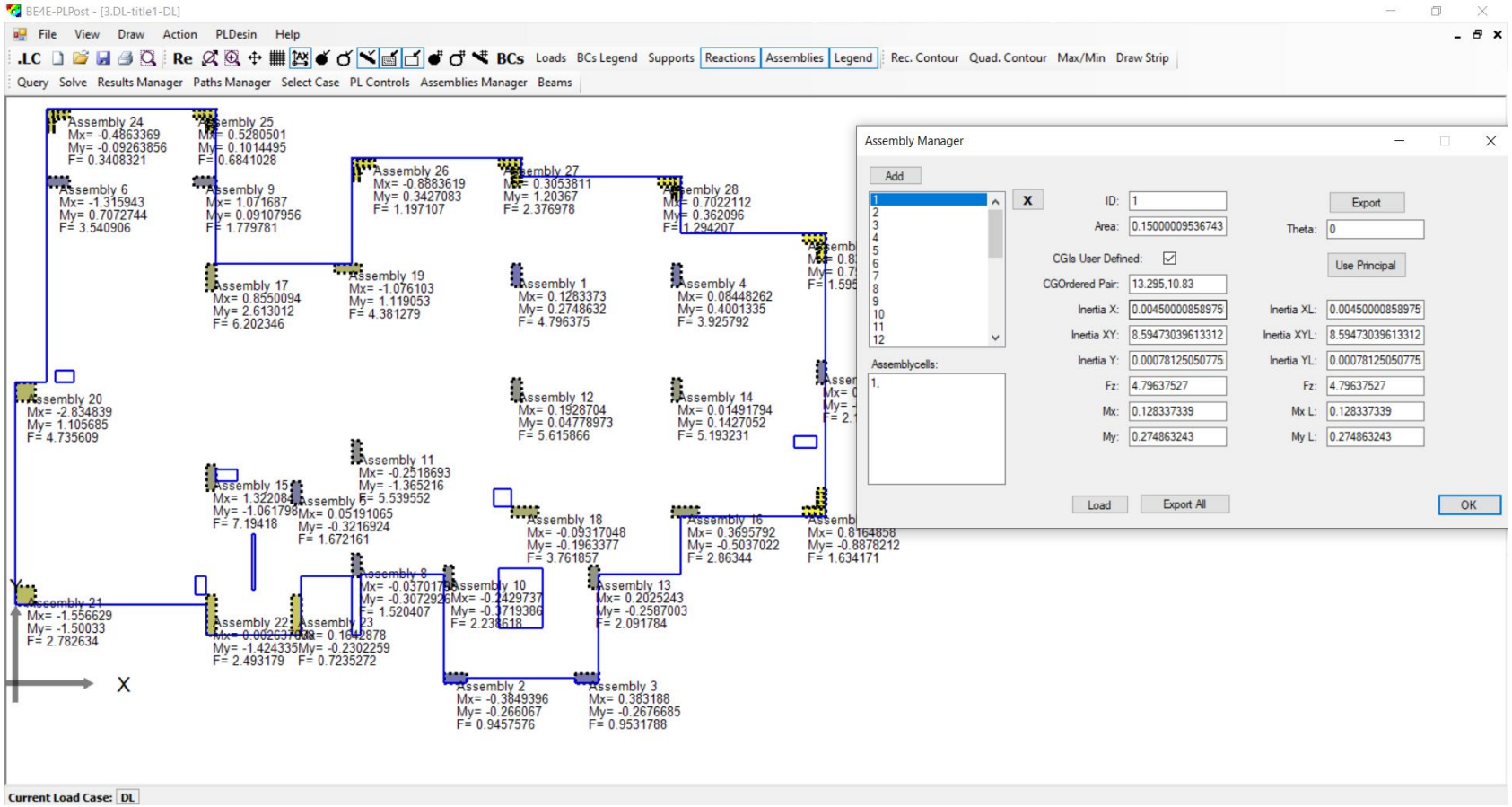
3. Slab results - supporting elements results

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

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

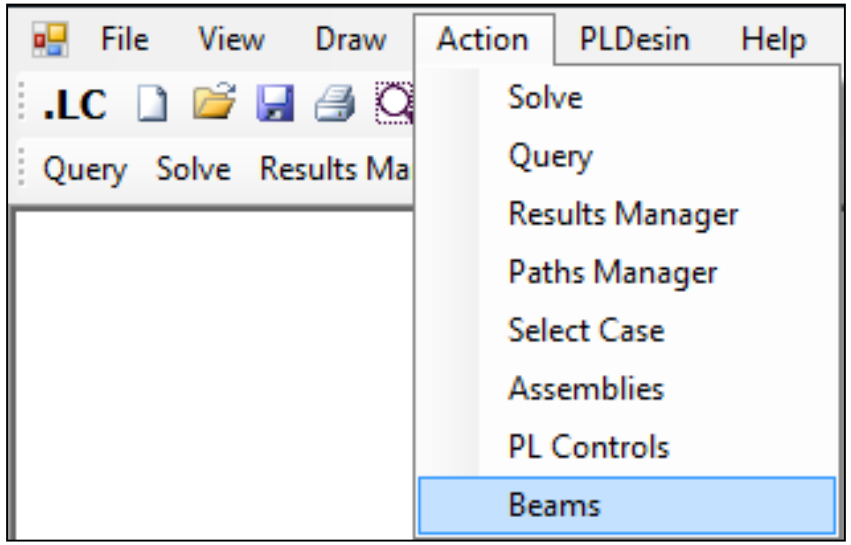


OR



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.

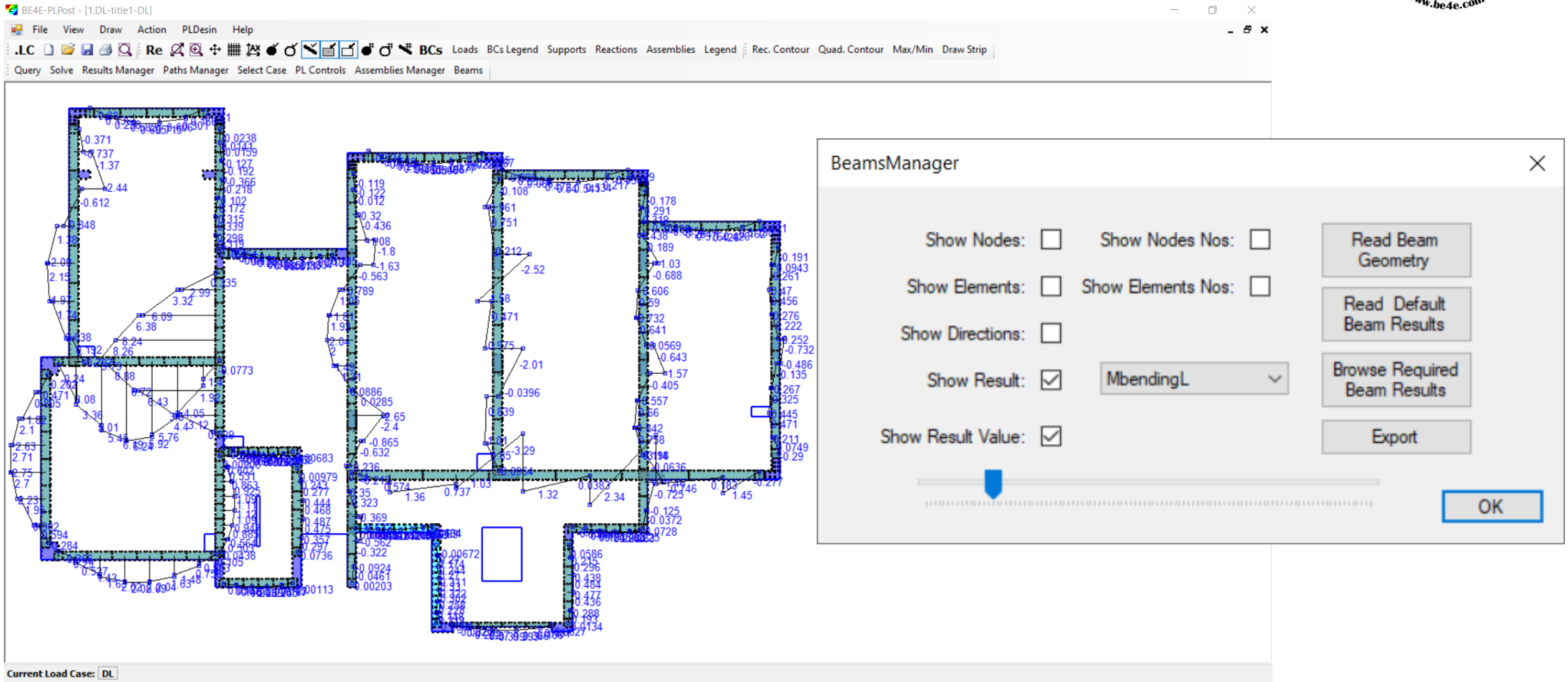


OR

CUFE-BE

www.be4e.com

4. Beam results



CUFE-BE

www.be4e.com

Table of contents

CUFE-BE

www.be4e.com

Table of contents

6. 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.

6. 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

Table of contents

CUFE-BE

www.be4e.com

Table of contents

7. Conclusions:

By the end of this lecture the attendee learned how to:

1. **Model practical slabs using PLGen.** ✓
2. **Model raft over Winkler area springs using PLGen.** ✓
3. **Check the numerical model using PLView.** ✓
4. **Run the model using PLCoreMan.** ✓
5. **Show results – Strip / contours / reactions / assemblies results using PLPost.** ✓
6. **Show beam results using PLPost.** ✓
7. **Learn PLPAK modelling capabilities.** ✓