



PLPAK – Basic Package





Disclaimer

Considerable time, effort and expense have gone into the development and documentation of the PLPAKTM software. The PLPAKTM software has been thoroughly tested and used. The PLPAKTM software should be used by engineers with good understanding of concrete behavior, pre-stressing and structural mechanics. The user accepts and understands that no warranty is expressed or implied by the developers or the distributors on the accuracy or the reliability of the PLPAKTM software. The user must explicitly understand the assumptions of the PLPAKTM software and must independently verify the results produced by the PLPAKTM software.

Copyright

Copyright © BE4E.com, 2000-2022

All rights reserved.

The PLPAKTM, PLGenTM, PLViewTM, PLCoreManTM, PLPostTM, PTPAKTM, PLTM, PL.EXETM are registered trademarks of BE4E.com.

The computer program PLPAKTM and all associated documentation are proprietary and copyrighted products. Worldwide rights of ownership rest with BE4E.com.

Unlicensed use of these programs or reproduction of documentation in any form, without prior written authorization from BE4E.com is explicitly prohibited.

No part of this publication may be reproduced or distributed in any form or by any means, or stored in a database or retrieval system, without the prior explicit written permission of the BE4E.com.



Table of contents

- 1. Introduction
- 2. PLGen Model generator
- 3. PLView Numerical model
- 4. PLCoreMan Manager and solver
- 5. PLPost Post processing
- 6. PLPAK modelling capabilities
- 7. Conclusions





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.





Table of contents

1. Introduction

- 2. PLGen Model generator
- **3.** PLView Numerical model
- 4. PLCoreMan Manager and solver
- 5. PLPost Post processing
- 6. PLPAK modelling capabilities
- 7. 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.











PLPAK Basic – package components:

- 1- Generator (PLGen).
- 2- Boundary element numerical model viewer (PLView).



3- Manager (PLCoreMan).



4- Post-processing (PLPost).





Table of contents

- 1. Introduction
- 2. PLGen Model generator
- **3.** PLView Numerical model
- 4. PLCoreMan Manager and solver
- 5. PLPost Post processing
- 6. PLPAK modelling capabilities
- 7. Conclusions





Table of contents

- 1. Introduction
- 2. PLGen Model generator
- 3. PLView Numerical model
- 4. PLCoreMan Manager and solver
- 5. PLPost Post processing
- 6. PLPAK modelling capabilities
- 7. Conclusions







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



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



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.







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.

Materials Load Cases Model Info Points table

Load Cases		
dead live fc	NEW EDIT DELETE	Loadcases for Own Weight (if any) Slab OW: dead Beam OW: dead OK Cancel
	New Load Case Enter Load Case Name machine	OK Cancel





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



• The user can change number of Gauss points to decrease time consumption in modeling but the number should be <u>even number</u>.

OR

• The user can change the type of solver either LU Decomposition, Gauss Elimination, or GPU solver.

• Also the user can change the PL Controls.





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 <u>Update \$PLCTRL\$.</u> button.
- Restore PLCTRL defaults button restore them to the default values.

			BEL 4 L	Ex and
PL Controls Manag	er (\$P	LCTRL\$.)	—	\times
PL Control No.	^	PL Control value	Update changes	
21 31 51 61 71 81 91 101 111 121		Description		^
121 131 151 161 171 181 191 201		Controls are loaded from \$PLCT	RL\$.	~
213 224 231 244	~	Update \$PLCTRL\$.	Restore PLCTRL defa	ults
24 4				

MENTS





The user can insert the model by three ways







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

File	View Tools	Define	Help		
	New .gen Ctrl	+N D	XF Re	CIr Ø €	
2	Open .gen Ctrl	+0 A	rray Match	h Wall Ass	
	Import	- • [DXF		
	Export	E I	Text fo	rmat	
B	Save .gen Ctr	I+S			
3	Print Ctrl	+ P			
	Print Preview				
	Page Setup				
	Exit Alt+	-F4			

OR







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

- 1- The slab line should be the outer line for the drawing.
- 2- All structural elements should be polyline in AutoCAD.
- 3- Beams should be inserted in AutoCAD is points at start/end of the beams.
- 4- All structural elements are drawn by four points only except the slab and the opening.







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.

Re Clr 🖉 🕀 🕂 🗮 🖄 Coordinates Divisions Loads 🗊 3D View





<i>4</i> 3	EMENTS	FOR
APART.	BE	Friend
Bou	4 E	EERS
	www.be4e.com	

View	w 1	Tools	Define	Help
	Too	lbars		•
	Grid	spaci	ng	
	Sho	w/Hid	e Loads	
	Sho	w/Hid	e Divisions	5
~	Sho	w/Hid	e Coordint	tes
~	Sho	w/Hid	e Grid	
~	Sho	w/Hid	e Axes	
	Win	dows		•
	3D (OpenG	L view	
	BEN	Aodel		







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.

L	DI Con modos		
End Mid Grid Nearest Points BE Model K Calc. Run	Select Edit Draw Segments Points Beams		





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.







Objects in the PLGen can be categorized into three categories:

- Slab and openings (Domain of the problem). \rightarrow Drawn using any number of points.
- Supporting elements. \rightarrow 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. \rightarrow 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.

Select Edit Draw Segments Points Beams













<u>Slab</u>

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.







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.







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.





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.









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).





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.

Del Dupl. 🗿 🧿 Move Copy Array Match Wall Assembly Load Assembly Grab CG

OR











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.







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

wall Assembly	properties			
Condition:	Below_Only	•	Height:	3.4
Material:	Concrete	•	Ndivisions	: 4
CGIs User	Defined:	CGO	rdered Pair:	4.539, 1.184 Explode
_				OK Cancel

Non-quadratic column in PLGen





Supporting elements (Beams)

- First change PLGen mode to Beams.
- End Mid Grid Nearest Points BE Model K Calc. Run Select Edit Draw Segments Points 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.







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).







Supporting elements (Beams)

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







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.







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.




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.









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.





Loading elements (Wall load assembly)

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







Other PLGen tools

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







Del Dupl. 🗿 😋 Move Copy Array Match Wall Assembly Load Assembly Grab CG

Del Dupl. 🕤 🔁 Move Copy Array Match Wall Assembly Load Assembly Grab CG



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.







Re Clr 🖉 🕀 🕂 🗱 🎝 Coordinates Divisions Loads 🗊 3D View



•



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.





3. Edit the boundary element divisions

- The default number of boundary elements is four. ٠
- Change PLGen mode to Segments mode. ٠
- 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. ٠





SegmentsDialog	\times
Boundary Condition: Free Number of divisions: 1 OK Cancel	~



Table of contents

- 1. Introduction
- 2. PLGen Model generator —
- 3. PLView Numerical model

ELEMENTS FOR BE 4 E White bete com

ENGINEER

- 1. Edit model information \checkmark
- 2. Build model and define its elements
- 3. Edit the boundary element divisions
- 4. PLCoreMan Manager and solver
- 5. PLPost Post processing
- 6. PLPAK modelling capabilities
- 7. Conclusions



Table of contents

- 1. Introduction
- 2. PLGen Model generator
- 3. PLView Numerical model
- 4. PLCoreMan Manager and solver
- 5. PLPost Post processing
- 6. PLPAK modelling capabilities
- 7. Conclusions





- 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.

"•] B	BE4E-P	PLView - [Project	t 1]	
	File	View Help		_
:		New Window	Ctrl+N	Re
		Import Input Fi	ile	
		Import AIP File		
		Save	Ctrl+S	
		Export DXF		
		Page Setup		
	4	Print Preview		
	3	Print	Ctrl+P	
		Exit		
1	_			

💀 File	View	Tools	Define	Help	
i 🗋 💕 🕻	Т	oolbars		•	
Del Dupl	G	irid spacir	ng		
	S	how/Hide	e Loads		1
	S	how/Hide	e Divisions		
	S	how/Hide	e Coordint	es	
	S	how/Hide	e Grid		
	✓ S	how/Hide	e Axes		
	V	Vindows		•	
	3	D OpenGl	L view		
	В	E Model			



From PLView

From PLGen



- 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

ELEMENTS

NGINEERS

BOUNDAR



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



• In PLView user can use these features.



ELEMENTS

BOUNDARY

FOR

ENGINEERS



BE model of slab showing nodes, and extreme points numbers



BE model showing BE number, directions, and normal directions

BE model showing boundary conditions



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

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



Tabulated values of the model





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





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.







Table of contents

- 1. Introduction
- 2. PLGen Model generator
- 3. PLView Numerical model 🗸
- 4. PLCoreMan Manager and solver
- 5. PLPost Post processing
- 6. PLPAK modelling capabilities
- 7. Conclusions





Table of contents

- 1. Introduction
- 2. PLGen Model generator
- **3.** PLView Numerical model
- 4. PLCoreMan Manager and solver
- 5. PLPost Post processing
- 6. PLPAK modelling capabilities
- 7. Conclusions





- 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

BE Model K Calc. Run







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.

File View	Run	Help						
dead		PLView (BE mesh editor tool)						
🛚 live		PT cable calculator						
/ fc		PTUpdate (Post-Tensioning tool)						
		AutoCAD exporter						
		AutoCAD extractor						
		EHSPAK						
		PL.EXE (command-line solver)						
		PLPost (post-processing tool)						
-Check previous		PLDesign (RC design tool)						
-Ensure that IRUNFlags in all .RUN files are 1 case 1: dead IRUNFlag=1 case 2: live IRUNFlag=1 case 3: fc IRUNFlag=1 Checking the existance of the \$run\$. in each load case folder case 1: dead has \$run\$.								
ase 3: fc has \$ru	uriș. n\$.							

File View Run Help Ive Itve fc
Image: dead Image: dead
✓ dead ✓ live ✓ fc ✓ fc
✓ live ✓ fc
I fc
Check previous solutions via the .STT files
Ensure that IRUNFlags in all .RUN files are 1 Solution for: fc is finished
case 1: dead IRUNFlag=1
Case 2: IVE IXUNHag=1 Teledaing die .co me to equip die .so me to
case 1: dead has \$run\$.
case 2: live has frunt.
-start uning the core command line for different load cases
load case no. 1: dead case 2: live IRUNFlag=1
The Sruns, is copied, run for case: dead will start Case 3: fc IRUINFlag=1 Case 3: fc IRUINFlag=1
Joad case no. 2: live
The \$run\$, is copied, run for case: live will start case 2 live has \$run\$.
Solution for: live is finished case 3: fc has \$run\$.



File View

Ready to start yo

-Ensure that IRU

case 2: live IRUNFrage

case 3: fc IRUNFlag=1

case 1: dead has \$run\$.

case 2: live has \$run\$.

case 3: fc has \$run\$.

case 1: dead IRU

-Check previous

✓ dead

☑ live

⊡ fc

Run Help

EHSPAK

P-PPAK





3- Open PL controls, manual, and about

E 6	BE4E-PLCoreMan C:\Users\Ahmed I	Fady\Desktop\	_	×
File	View Run Help			
G	Open (.LC) file			
6	Re-load (.LC) file			
6	(.IN) to (.LC) tool (IN2LC)			
	PL controls (\$PLCTRL\$.)			
	Exit			
Read Chec Ensu case Chec case case	y to start your analysis k previous solutions via the .STT files re that IRUNFlags in all .RUN files are 1: dead IRUNFlag=1 2: live IRUNFlag=1 3: fc IRUNFlag=1 king the existance of the \$run\$. in eac 1: dead has \$run\$. 2: live has \$run\$. 3: fc has \$run\$.	1 ch load case folder		







Table of contents

- 1. Introduction \checkmark
- 2. PLGen Model generator \checkmark
- 3. PLView Numerical model 🗸
- 4. PLCoreMan Manager and solver 🗸
- 5. PLPost Post processing
- 6. PLPAK modelling capabilities
- 7. Conclusions





Table of contents

- 1. Introduction
- 2. PLGen Model generator
- **3.** PLView Numerical model
- 4. PLCoreMan Manager and solver
- 5. PLPost Post processing
- 6. PLPAK modelling capabilities
- 7. Conclusions







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.



•	File	View To	ols Def	ine	Hel	p		
Ma		New .gen	Ctrl+N		Poin	ts table 🕴 🗋 📂 🛃 🚑		
D	2	Open .gen	Ctrl+0		Array	Match Wall Assembly		
_		Import		۲				
		Export		۲		BE files		
		Course Child			Assemblies Beam assemblies			
		Save .gen Ctrl+S						
	3	Print	Ctrl+P			Text format		
	<u></u>	Print Preview				1		
		Page Setup						
		Exit	Alt+F4					

	LOEN - [OEOI	neuyj		
🖳 File	View To	ols De	fine	Help
i Mi 🗋	New .gen	Ctrl+N		Points table 📋 🚰 🛃 🖪
i d 🞽	Open .gen	Ctrl+0		Array Match Wall Assembly I
	Import		۲	
	Export		•	BE files
	Save gen	Ctrl+S		Assemblies
	Save igen	Cuita		Beam assemblies
3	Print	Ctrl+P		Text format
	Print Previe	w		
	Page Setup			
	Exit	Alt+F4		
			-	1

DEAE DI Con (Coomotral)



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

Select Case			×	
DL LL Flooring Wall load Combination 1	Combination Name: Add Entry Remove Entry	Combination 1 Wall load ✓ DL LL Flooring Wall load	1.4 1.4 1.6 1.4 1.4	
ADD REMOVE			OK Cancel	



Current Load Case: DL





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

SE4E-PLPost - [1.DL-title1-DL]

🖳 File View	Draw Action PLDe	sin Help
.LC 🗋 💕 🛃	Linear Strip	i 🏞
Query Solve Re	Rec. Contour	Select
	Quad Contour	
	Principal Direction	s

1	BE4E-PLPost - [1.DL-title1-DL]													
	a 🚽	File	Viev	v Dra	w Actio	n PLI	Desin	Help						
1	.LO	C [) 📂	. 3	Q, Re	Ø	4	## 🏝	é đ	✓✓✓	e 4 ×	t BCs	Loads	BC
	Qu	iery	Solve	Results	Manager	Paths I	Manage	r Select	t Case	PL Controls	Assemblies	Manager	Beams	;

Draw quadratic contour

Rec. Contour Quad. Contour Max/Min Draw Strip

SE4E-PLPost - [1.DL-title1-DL]

🖳 File View	Draw	Action	PLDesin	Help
.LC 🗋 💕 🚽	Li	near Strip		I îA\$
Query Solve Re	R	Selec		
	Q			
	P	rincipal Dir	ections	

Draw strip

Rec. Contour Quad. Contour Max/Min Draw Strip

😪 BE4E-PLPost - [1.DL-title1-DL]







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.

.LC

- 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.

Query Solve Results Manager Paths Manager Select Case PL Controls Assemblies Manager Beams

Draw

View

Query Solve Results Ma

Action

Solve

Query



OR

PLDesin

Results Manager

Paths Manager

Select Case

Assemblies PL Controls

Beams

Help

Case



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.

Re 🖉 🏵 🕂 🗰 🖄 🗉 🗹 🎽 💣 🗗 🔻 BCs Loads BCs Legend Supports Reactions Assemblies Legend

SE4E-PLPost - [1.DL-title1-DL]

.LC 🗋 📂 🛃 🎒 🔾

Query Solve Results Ma

OR

File

View Draw

Action

Solve

Query

Results Manager

Paths Manager

Select Case Assemblies

PL Controls Beams

PLDesin Help



Re 🖉 🕀 🕂 🗮 🎇 🗉 🗹 🎽 📹 🗗 💣 🗗 🤻 BCs Loads BCs Legend Supports Reactions Assemblies Legend



Query Solve Results Manager Paths Manager Select Case PL Controls Assemblies Manager Beams

FLEMENTS

www.be4e.cov

BOUNDARY

FOR

ENGINEERS





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.







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.
 File View Draw Action PLDesin Help
- 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.

Results Manager	×			
Strips				
	X ID: Enabled: Export	and a standard and a		
	Npoints: Result:			
	Start:			
	Status:			
Contours Main Contour	ID: Contour 1 Enabled: I Export			
Contour 1	N: 15 Spacing: 0.1			
	Min le leardafinad:min:			
	Max Is Userdefined: max: 0			
	Status: ToBeSolved Current Variable: Mox ~			
l l	Itheta: 0			
Column Plot		L	and the second se	
Enabled:	N: 8 Max Is Userdefined: max: 0			
0	Min le lleerdefined:min: 0			
Current variable				
R	afresh Show Total Forces: Export			

Current Load Case: DL





SolverDialog

Status Log

Solution completed.

Carrying out file system check. Reading the basic input. Reading the rest of the Input. Carrying out internal checks. Forming Gauss arrays. Reading the internal points data. Computing internal point stress resultants.



n x

3. Slab results – contour results

Straining action for rectangular contour using result manager:

• Solve the main contour.

Internal points displacements: 3894 of 3894 Completed. Internal points stress resultants: 3894 of 3894 Completed.

• Solver dialog will appear and after solving press on (F3) or refresh.

BF4F-PLPost - [2.DL-title1-DL]

• The user can show/hide the legend.

nde the legend.	File View Draw Action PLDesin Help I.C D D D A Contour PLDesin Help I.C D D D A Contour Quad. Contour Max/Min Draw Strip Query Solve Results Manager Paths Manager Select Case PL Controls Assemblies Manager Beams
X	I set the set of th
	Current Load Case: DL




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.

Results Manager			×				
Strips							
	X ID:	Enabled: Export					
	Npoints:	Result: V		9.1J1	Sector Construction		
	Start:	End:					
	Status:	v theta:					
				1			
						H	
Contours	0					-	
Main Contour Contour 1	X ID: Quad I	Enabled: 🗹 Export				H	
Quad 1	N: 15	Spacing: 0.2		*			
	Min Is Userdefined:	min: 0				-	
	Max Is Userdefined:	max: 0		/			
	Status: ToBeSolved	Current Variable: Mxx ~					
	Itheta: 0						
Column Plot				L .			
Enabled:	N: 8 Max Is Userde	efined: max: 0			-		
Current Variab	le: FZ V Min Is Userde	efined: min: 0		:			
	Show Total For	rces: Export					
	heiresn				-		
		Close					





n ×

_ 8 ×

- 0.329 - 0.222 - 0.115 - 0.00725 - 0.10725 - 0.207 - 0.315 - 0.422 - 0.529 - 0.636 - 0.744 - 0.851 - 0.951 - 0.951 - 1.17

Mxx

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.

[(
	Query Solve Results Manager Paths Manager Select Case PL Controls Assemblies Manager Beams
Internal points displacements: 2267 of 2267 Completed. Internal points stress resultants: 2267 of 2267 Completed. Status Log Carrying out file system check. Reading the basic input. Reading the rest of the Input. Carrying Gauss arrays. Reading the internal points data. Computing internal point stress resultants. Solution completed.	Cuery Solve Results Manager Paths Manager Select Case PL Controls Assemblies Manager Beams

Draw Action PLDesin Help

😴 BE4E-PLPost - [2.DL-title1-DL]

View

Current Load Case: DL





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. ٠
- The user doesn't need to go to Result Manager as any previous Contour. •

SolverDialog	×
Internal points displacements: 1 of 1 Completed.	
Internal points stress resultants: 1 of 1 Completed.	
Status Log	
Canying out file system check. Reading the basic input. Reading the rest of the Input. Canying out internal checks. Forming Gauss arrays. Reading the internal points data. Computing internal point stress resultants. Solution completed.	
OK Cancel	

							_
Monitor						×	
Loadcase DL LL Flooring Wall load	X 2.87384 2.87384 2.87384 2.87384	Y 11.86531 11.86531 11.86531 11.86531	Rx -2.8171E-05 4.5301E-06 3.6241E-06 1.8121E-06	Ry -0.00017283 -8.4156E-05 -6.7325E-05 -3.3662E-05	Uz -0.00074689 -0.00045005 -0.00036004 -0.00018002	Mxx -0.25 -0.15 -0.07	
<						>	
						lose	





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.

nternal points displacements: 195 of 1	195 Completed.	
nternal points stress resultants: 195 of	f 195 Completed.	
tatus Loo		
Carrying out file system check.		
reading the basic input. Reading the rest of the Input.		
Carrying out internal checks. Forming Gauss arrays.		
Reading the internal points data. Computing internal point stress resulta	ants.	
Solution completed.		



Current Load Case: DL





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. ٠
- 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.

	BE4E-F	LGen - [Geome	etryj							ъ В	E4E-PLPC	ost - [3.L	L-title	I-DL]			
	File	View Tool	ls Define		Help					•	File	/iew	Draw	Actio	n PLDesin	Help	
÷ M		New .gen	Ctrl+N	P	oints table 🕴 🗋 🚰 🛃 🚑					÷.ь	с 🗋 🛛	i 🛃 🗧	3 Q	1	Solve		k
i D		Open .gen	Ctrl+0	Ar	ray Match Wall Assembly					Qu	ery Solv	/e Resu	ults Ma	1	Query		c
		Import	•								-				Results Manag	jer	ł
		Export	•		BE files									1	Paths Manage	r	I
		Save .gen	Ctrl+S		Assemblies										Select Case		L
				-	Beam assemblies										A 11		ıl.
		Print	Ctrl+P		Text format										Assemblies		4
		Print Preview													PL Controls		L
		Page Setup							OR						Beams		L
		Fxit	Alt+F4							I				_			1
						Ouen	Solve	Results Manager	Paths Manager Sele	rt Cas	e DIC	ontrol	Δεε	embli	es Manager	Ream	-





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.





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.



ELEMENTS

FOR

OR

Query Solve Results Manager Paths Manager Select Case PL Controls Assemblies Manager Beams









- 1. Introduction \checkmark
- 2. PLGen Model generator \checkmark
- 3. PLView Numerical model 🗸
- 4. PLCoreMan Manager and solver 🗸
- 5. PLPost Post processing \checkmark
- 6. PLPAK modelling capabilities
- 7. Conclusions





- 1. Introduction
- 2. PLGen Model generator
- **3.** PLView Numerical model
- 4. PLCoreMan Manager and solver
- 5. PLPost Post processing
- 6. PLPAK modelling capabilities
- 7. Conclusions





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.



- 1. Introduction
- 2. PLGen Model generator \checkmark
- 3. PLView Numerical model 🗸
- 4. PLCoreMan Manager and solver 🗸
- 5. PLPost Post processing \checkmark
- 6. PLPAK modelling capabilities 🗸
- 7. Conclusions





- 1. Introduction
- 2. PLGen Model generator
- **3.** PLView Numerical model
- 4. PLCoreMan Manager and solver
- 5. PLPost Post processing
- 6. PLPAK modelling capabilities

7. Conclusions





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.







For additional help please visit: <u>https://www.plpak.com</u>

> or send e-mail to: plpak@be4e.com