

The Fixed Base Package

User MANUAL

PLPAKTM Version 2.00 STRUCTURAL ANALYSIS SOFTWARE USING THE BOUNDARY ELEMENTS METHOD

Copyright © 2000-2022 <u>https://www.plpak.com</u> E-mail: <u>support@plpak.com</u>

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, PL.exeTM, PLDesignTM, TBPAKTM 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.

Further information and copies of this documentation may be obtained from:

<u>Technical director:</u> Youssef F. Rashed, PhD Department of structural engineering, Cairo University, Egypt.

e-mail: <u>plpak@be4e.com</u> web: <u>https://www.plpak.com</u>

Introduction:

In this manual, the user should know how to start his lateral model from Autodesk Revit until finishing his model reinforcement details in Autodesk Revit.

Therefore, this manual will explain the following:

- Modeling into Autodesk Revit including all precautions.
- Solving lateral analysis in TBPAK.exe
- Showing straining actions in PLPost.exe
- Designing slabs, beams and check punching in columns in PLdesign.exe
- Displaying the deformed shape for the structure due to different types of load or different types of load combination in OpenGL.
- Showing the straining actions and design the vertical element in OpenGL.
- Importing the reinforcement into Autodesk Revit.

A- Modeling into Autodesk Revit:

<u>First</u>, the user should choose the suitable template into Autodesk Revit before starting his model; this template is called Structural Analysis-DefaultMetric-Up.rte as shown in figure (1). The user can download it from internet.

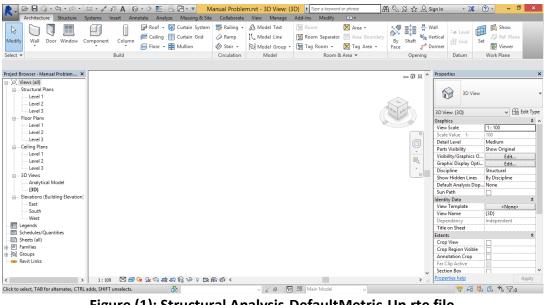


Figure (1): Structural Analysis-DefaultMetric-Up.rte file

To open this template:

New ----> Project ----> Browse ----> Structural Analysis-DefaultMetric-Up.rte

Second, the user should insure that the computer is capable to create files in the PLPAK folder in the setup place.

To insure this step:

eneral Sharing	Security	Previous Vers	ione Cue	tomize	
	g occounty	Trevious vers		comize	
Object name:	C:\Program	Files\PLPAK			
Group or user n	ames:				
	ICATION PA	CKAGES			~
		PLICATION PA	CKAGES		
REATOR	OWNER				
					~
< .				>	
To change per	nissions, clic	k Edit.		Edit	
Permissions for				_	
APPLICATION	PACKAGES		Allow	Deny	
Full control			\checkmark		^
Modify			\checkmark		
Read & exec			\checkmark		
List folder cor	ntents		\checkmark		
Read			\checkmark		
Write			\checkmark		Y
For special pem		dvanced setting	s. ,	Advanced	
click Advanced					

Figure (2): PLPAK Properties folder

<u>Third</u>, the user should **insure that the whole structure is in the positive quarter** as shown in figure (3) by either drawing from the Autodesk Revit or importing the .dxf files from AutoCAD.

To show the original point in Autodesk Revit:

View → Visibility/Graphics → Model Categories → Site → Survey Point

	F	Projection/Surface C			ıt		Detail	^
Visibility	Lines	Patterns	Transparency	Lines	Patterns	Halftone	Level	
Raster Images							By View	+
• Roads							By View	
Roofs							By View	
Security Devices							By View	
							By View	
General Site							By View	
Hidden Lines								
Pads								
Project Base Point								
Property Lines		1						
Survey Point								
. Specialty Equipment							By View	
Sprinklers							By View	
+ Stairs							By View	
Structural Area Reinf							By View	
· · · ·		-	li la					
All None	Inve	rt Exp	oand All	erride Host Laye	rs			
	-			Cut Line St <u>y</u> les			<u>E</u> dit	

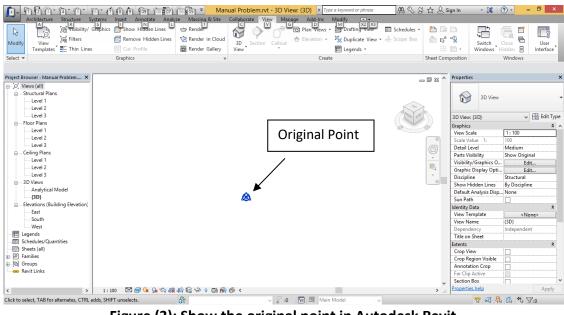


Figure (3): Show the original point in Autodesk Revit

Fourth, the user should adjust both project units and material for structural members,

To adjust the project units in kN.m as shown in figure (4) in Autodesk Revit: Manage → Project Units → Common → Length in meters Structural → Force in kN

	Project	t Units	×	Proje	ct Units	
iscipline:	Common			Discipline: Structural		
Units	s	Format	<u>^</u>	Units	Format	
Length		1234.568 [m]		Force	1234.57 kN	
Area		1234.568 m ²		Linear Force	1234.57 kN/m	
Volume		1234.57 m ³		Area Force	1234.57 kN/m ²	
Angle		12.35°		Moment	1234.57 kN-m	
Slope		12.35°		Linear Moment	1234.57 kN-m/m	
Currency		1234.57		Stress	1234.6 MPa	
Mass Density		1234.57 kg/m³		Unit Weight	1234.6 kN/m³	
				Weight	1234.57 kN	
				Mass	1234.57 kg	
				Mass per Unit Area	1234.57 kg/m ²	
				Thermal Expansion Coefficie	1234.56789 1/°C	
				Point Spring Coefficient	1234.6 kN/m	
				Line Spring Coefficient	1234.6 kN/m ²	
				Area Spring Coefficient	1234.6 kN/m³	
				Datational Datat Casima Case	1004 CTNL /8	F
Decimal symbol/d	digit <u>a</u> rouping:			Decimal symbol/digit grouping:		
123,456,789.00	•			123,456,789.00 ¥		
	ОК	Cancel <u>H</u> elp		ОК	Cancel He	elp

To adjust the material properties for structural floors, beams and columns as shown in figure (5) in Autodesk Revit:

Manage → Object Styles → Model Object → Floors → Material (Concrete, Cast in Situ)

_	Line	Weight				
Category	Projection	Cut	Line Color	Line Pattern	Material	
+ Flex Pipes	1		Black	Solid		
Floors	1	4	Black	Solid	Concrete, Cast In Si	
Furniture	1		Black	Solid		
Furniture Systems	1		Black	Solid		
Generic Models	1	1	Black	Solid		
HVAC Zones	1		Black			
Lighting Devices	1		Black			
Lighting Fixtures	1		Black	Solid		
Mass	1	2	Black	Solid	Default Form	
Mechanical Equipment	1		Black	Solid		
Nurse Call Devices	1		Black			
Parking	1		Black	Solid		
Parts	1	2	Black			
Pipe Accessories	1		Black	Solid		
Pipe Fittings	1		Black	Solid		
Pipe Insulations	1		Black	Solid		
Pine Placeholders	1		RGR 000-127-000			
Select All Select Nor	ne <u>I</u> nvert	:	Mod	dify Subcategories	elete <u>R</u> ename	

Figure (5): choose the material properties for slabs, beams and columns

To change the mechanical properties of concrete double click on Concrete, Cast in Situ then the material window is open, choose the mechanical list under the physical properties icon as shown in figure (6).

Material B	rowser - Concrete, Cast In Situ	?	×
Search Q	Identity Graphics Appearance P	'hysical Thermal	
Project Materials: All 🔹	Concrete 10 MPa		<u>`` ×</u>
Name	▶ Information		
Brick, Common	Basic Thermal Thermal Expansion Coefficient	0.00001 inv °C	÷
Cavity Fill	▼ Mechanical		
Concrete - Cast-in-Place Concrete - 35 MPa		Isotropic	*
	Young's Modulus Poisson's Ratio		÷ ÷
Concrete Masonry Units	Shear Modulus		÷
Concrete, Cast In Situ		2,406.45 kg/m ³	¢
Concrete, Cast-in-Place gray	► Concrete		
Concrete, Sand/Cement Screed			
Copper			
Damp-proofing			
🛐 • 🚇 • 🗏 🛛 🔍			
Ø		OK Cancel	Apply

Figure (6): Material Browser window

In case of existing of shear walls or cores in the structural building, the user should adjust the material properties of shear walls and cores as shown in figure (7)

Manage → Object Styles → Model Object → Walls → Material (Concrete, Cast in Situ)

	Line	Weight			
Category	Projection	Cut	Line Color	Line Pattern	Material
Structural Beam Systems	6		PANTONE Process	Dash	
Structural Columns	1	4	Black	Solid	
Structural Connections	1	1	Black	Solid	
Structural Fabric Areas	1	1	Black	Solid	
Structural Fabric Reinforce	1	1	Black	Solid	
Structural Foundations	2	4	Black	Solid	
Structural Framing	1	4	Black	Solid	
Structural Path Reinforce	1	1	Black	Solid	
Structural Rebar	1	1	Black	Solid	
Structural Stiffeners	1	1	Black	Solid	
Structural Trusses	6		PANTONE Process	Dash	
Telephone Devices	1		Black		
Topography	1	6	Black	Solid	Earth
🖬 Walls	2	4	Black	Solid	Concrete, Cast In Situ
Windows	2	2	Black	Solid	
. Wires	1		Black	Solid	
Select All Select None	Invert		Modi	fy Subcategories	<u>D</u> elete <u>R</u> ename

Figure (7): choose the material properties for walls

Fifth, the model can be drawn by two options:

- I- Autodesk Revit as shown in figure (8)
- II- Imported from .dxf AutoCAD file as shown in figure (9).

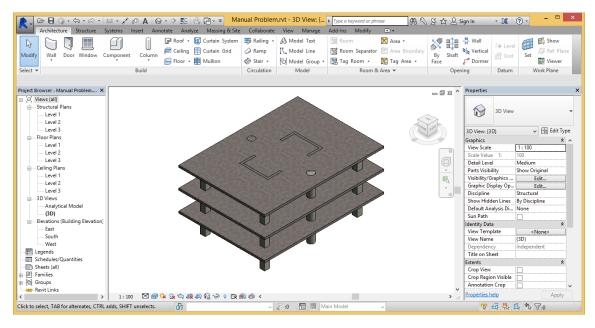
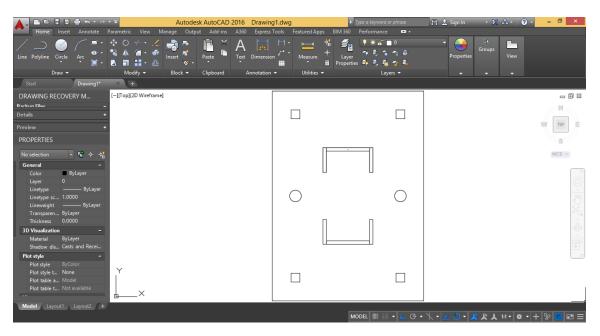


Figure (8): Autodesk Revit structural model





I- Draw from Autodesk Revit:

1- Draw the Grids as shown in figure (10) Structure → Grid

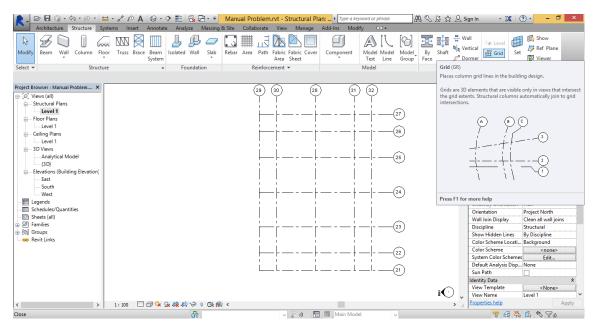


Figure (10): Grid Drawing

2- Draw the slab with the required properties as shown in figure (11)
 Structure → Floor → Floor Structural

R - D - G - G - C - H - N D A D - 9 H - R			- 0 ×
Architecture Structure Systems Inset Annotate Analyze Massin Modify Select + Properties Clipboard Geometry Modify	🖾 🚽 🚔 🗶 🍈 🎽 📔 📩 🚺 👘 🖓 🖥 Boundary	Highlight Modify Sub Elements S Pick Supports	Path Fabric Fabric Area Sheet
Modify Floors			
Project Browser - Manual Problem × □·	29 30 28 31 32 I I I I I I I I I I I I I I I I I I I	Properties	×
Level 1		Floors (1)	🗸 🔐 Edit Type
i⊐ – Ceiling Plans ↓ _ Level 1 m → 3D Views			evel 1
Analytical Model (3D)		Height Offset From 0. Room Bounding Related to Mass	
Elevations (Building Elevation) East South		Structural Structural Enable Analytical M	
- West - El Legends 		Rebar Cover - Top F In Rebar Cover - Botto In	nterior (slabs, wall nterior (slabs, wall
Sheets (all)		Rebar Cover - Other In Dimensions	nterior (slabs, wall
Families		Slope	×
⊕[@] Groups			2.0000
🚥 🚥 Revit Links		Area 2	34.000 m²
	i i i	Volume 70	0.200 m ²
		Thickness 0.	.3000
		Identity Data	\$
		Image	
	21	Comments	
		Mark	
		Phasing Properties help	× ×
< > 1:100 🗆 🗇 🔽 🚓 🚓 🖓 🔗 9 🖽 f		/] — · · · · · ·	Apply
Click to select, TAB for alternates, CTRL adds, SHIFT unselects.	v 🖉 :0 🔚 💹 Main Model	v 🤨 🕰 🛼 (🕹 🏞 🖓:1

Figure (11): Slab Drawing

Please check that there is not zero level as shown in figure (12) and check also that the model in the **positive quarter** as shown in figure (13).

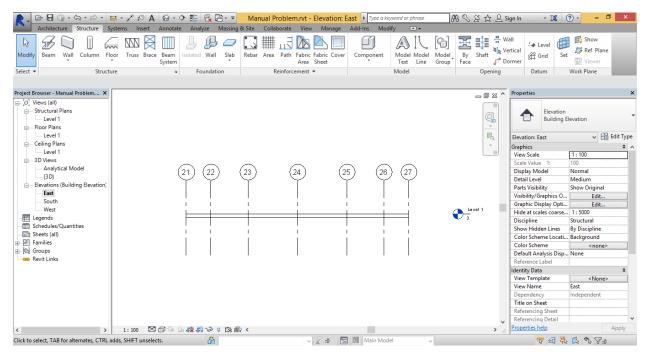


Figure (12): Check zero level

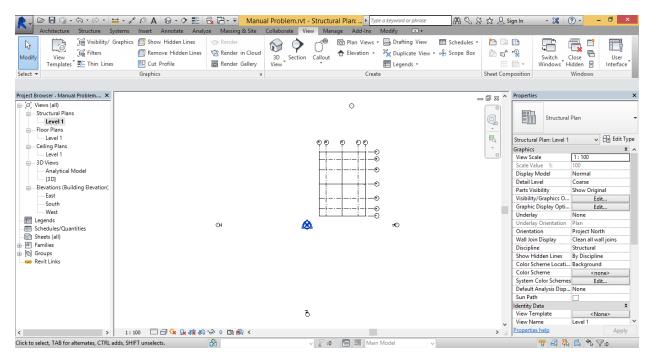
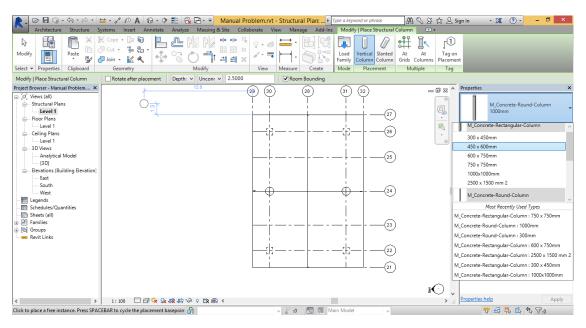


Figure (13): Check the model is in positive quarter

3- Draw columns (either circle or quadratic), then adjust its properties as shown in figure (14).



Structure ---- Columns

Figure (14): Columns drawing

4- Draw cores and shear walls, then adjust its properties as shown in figure (15).
 Structure → Walls → Walls: Structural

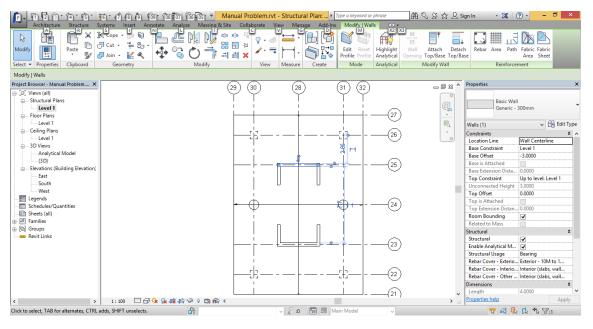


Figure (15): Cores and shear walls drawing

5- Draw beams and then adjust its properties as shown in figure (16).
 Structure → Beams

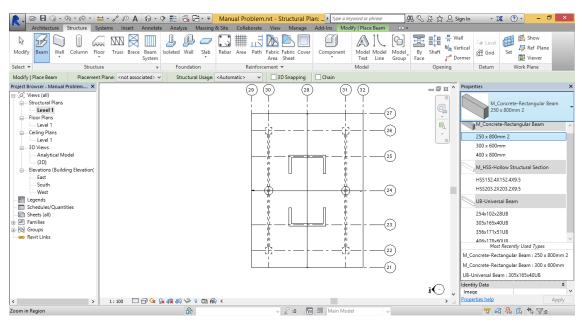


Figure (16): Beams drawing

6- Insert different level to replicate the slab to model the whole building as shown in figure (17).
 Structure → Levels

🜔 🕞 🖯 🎯 • 🗞 • 🔊 • 😫	= · 🖍 🕫 A 🞯 · 🕈 🏗 🗟 🖻 · 🔻	Manual Problem.rvt - Elevation: E	ast Type a keyword or phrase	AA 🔍)	조 요 Sign In 🔹 🕱 🕜 - 🗖 🗙
Architecture Structure Sy	stems Insert Annotate Analyze Massing	& Site Collaborate View Manage	Add-Ins Modify 🔿 🕶		
Modify Beam Wall Column Fig. Select Structure	System	Rebar Area Path Fabric Fabric Cover Area Sheet		Model By Group Face	Level (L) Places a level in a view.
Project Browser - Manual Problem ×					Levels are 3D elements that are only visible in views that intersect the level extents. Most building elements, such as floors and beams, are hosted by levels. Other elements, such as columns and walls, are constrained to levels.
Level 1 Level 2 Level 3 Floor Plans					
Level 1 Level 2 Level 3 Ceiling Plans	<u>● lemi3</u> —				
Level 1 Level 2 Level 3 D Views	<u>€ 160012</u> —			-	Press F1 for more help
Analytical Model {3D} Elevations (Building Elevation)		3	<u>26</u> −27 €,	nel 1	Discipline Structural Show Hidden Lines By Discipline Color Scheme Locati Background
South					Color Scheme <pre></pre> Color Scheme <pre></pre> Color Scheme <pre></pre> Color Scheme <pre></pre> Color Scheme Color Scheme Color Scheme <pre></pre> Color
Egends Schedules/Quantities					Identity Data 🎗
					View Template <none></none>
■ P Families					View Name East Dependency Independent
Groups					Title on Sheet
😡 Revit Links					Referencing Sheet
					Referencing Detail
< >	1:100 🔳 🗐 🔆 🔅 👼 🏟 や 🖇 🛤	ζ γ <			Properties help Apply
Click to select, TAB for alternates, CTRL a			Main Model 🗸		♥ ഛ ♣ ₲ ₺ ♥:0

Figure (17): Determine levels

7- Copy the drawn level 1 into the created levels as shown from figure (18) to (20).

Select the structural members → Copy to Clipboard → Paste → Aligned to Selected Levels

🌔 🕞 🖯 🎯 • 🖘 🕫	- 🖴 - 🖍 🕫 A 🞯 - 🌩 🏗 🗟 🗟 - 🔻 Manual F	Problem.rvt - Elevation: East 🕨 🔈	pe a keyword or phrase	舟 🔍 & ☆ 🚨 Sign In	- 🛛 🖓 - 🗖 🗡
Architecture Structure	Systems Insert Annotate Analyze Massing & Site Colli	aborate View Manage Add-Ins	Modify Multi-Select	•	
Modify Select - Properties Clipboard Modify Multi-Select Activ	Copy to Clipboard Copies selected elements to the clipboard.	♀ · ∅ ↓ ↓ ↓ ↓ ↓ ✓ · ≒ ↓ ↓ ↓ ↓ ↓ View Measure Create	Filter		
Project Browser - Manual Problem □ [0] Views (all) □ Structural Plans □ Level 1 □ Level 2 □ Level 3 □ - Floor Plans □ Level 1 □ Level 2 □ Level 3 □ - Floor Plans	After copying elements to the clipboard, use a Paste tool to paste the copied fements in the current view, a different view, or another project. Clipboard panel > Copy is different from Modify panel > Copy. Use Modify panel > Copy when you want to copy selected elements and place them immediately (for example, when you need to switch views before placing the copied elements. Press 11 for more help				Multiple Categories Selected * mmon (22) * 🛱 Edit Type
Celling Plans Level 1 Level 2 Level 2 Analytical Model (3D) Elevations (Building Elevat South West West	on]		27) •		
Schedules/Quantities Schedules/Quantities	> 1:100 回回信 () 液液的や 9 四倍 < TRL adds, SHIFT uncelects. 作	v 2₂:0 ि ∭ Ma	n Model v	> _{at} Prov	perties.help Apply 電気路氏へ下空22

Figure (18): Select and Copy drawn level

	😫 - 💉 🕫 A 😡 - 🔿 🖭 🗟 🖓 Manual Problem.rvt - Elevation: East 🕨 Type a keyword or phrase 🥅 🕅 Systems Insert Annotate Analyze Massing & Site Collaborate View Manage Add-Ins Modify (Multi-Select 🔾 -	김 🔍 윤 ☆ 🗘 Sign In 🛛 🔹 🗶 🕘 -	□ ×
Nodify Paste Select • Properties	Appleting Indext Animodals Animoda		
Modify Multi-Select	to Selected Levels	- X Properties	×
Co, Views (all) Structural Plan Level 1 Level 2	Aligned to Selected Levels	Multiple Categories Selec	cted 👻
Evel 3 Evel 3	to Same Place	Common (22)	Edit Type
Level 3	to Picked Level		
Level 1 Level 2 Level 3	€ tayl 2 6		
Analytical Model (3D) Elevations (Building Elevation) East South West			
Schedules/Quantities Schedules/Quantities Sheets (all) Families Go Groups Revit Links			
< >>	1:100 🖻 母 添 品 森 心 9 時 前 <	> Properties help	Apply
Click to select, TAB for alternates, CTRL	Ladds, SHIFT unselects. 😽 🗸 🖓 🖬 Main Model 🗸	🎌 🖧 🛼 🖒 🏞 🖓:	.22

Figure (19): Paste the copied level on the selected levels

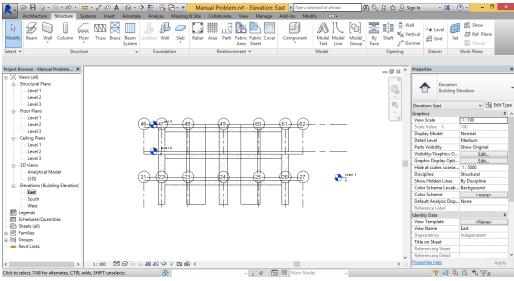


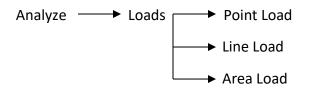
Figure (20): Modeling the whole building

8- Insert load cases and different types of loads (vertical/lateral)
 To define load cases as shown in figure (21):
 Analyze → Load Cases

Joine I	Representation Settings	Load Cases	Load Com	binations	Analytical M	lodel Settings	Boundary Cond	litions Setting	gs
oad <u>O</u>	Cases								
	Name	Case I	Number		Nature		Category	^	Add
1	SEIS1	8		Seismic		Seismic	Loads		
2	D.L	1		Dead		Dead Lo	ads		Delete
								~	
								*	
oad <u>N</u>	Natures							*	
oad <u>N</u>	latures			Name				×	Ad <u>d</u>
0ad <u>N</u>	Jatures			Name					_
				Name					Ad <u>d</u> D <u>e</u> lete
1	Dead			Name					_
1	Dead Live			Name					_
1 2 3	Dead Live Wind			Name					_
1 2 3 4	Dead Live Wind Snow			Name					_
1 2 3 4 5	Dead Live Wind Snow Roof Live			Name					_
1 2 3 4 5 6	Dead Live Wind Snow Roof Live Accidental			Name					_
1 2 3 4 5 6 7	Dead Live Wind Snow Roof Live Accidental Temperature			Name					_
1 2 3 4 5 6	Dead Live Wind Snow Roof Live Accidental			Name					_
1 2 3 4 5 6 7	Dead Live Wind Snow Roof Live Accidental Temperature			Name					_

Figure (21): Load Cases determination

To insert loads at each load cases as shown in figure (22):



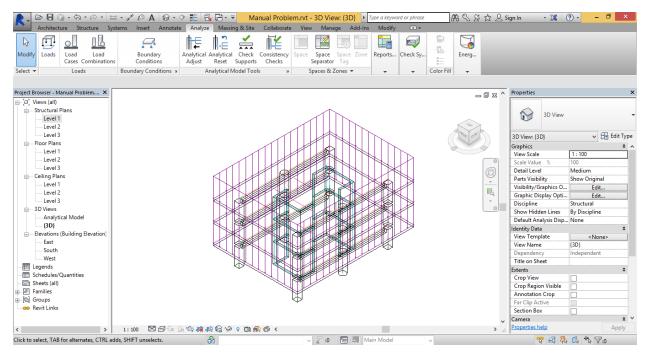


Figure (22): Loads in different load cases

II- Importing from .dxf AutoCAD file

In case of importing from .dxf file as shown in figure (23), the user should check on the scale of the drawings and the location of the imported level from the origin.

To import .dxf file in Autodesk Revit: Insert → Import CAD → select .dxf file

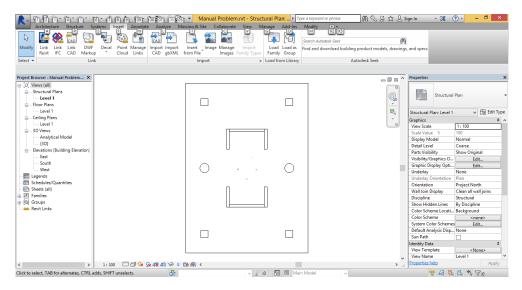


Figure (23): Import .dxf file in Autodesk Revit

Now the user should check the scale of the imported model as shown in figure (24):

To check the scale:

Modify — Measure between Two References — Scale — Numerical

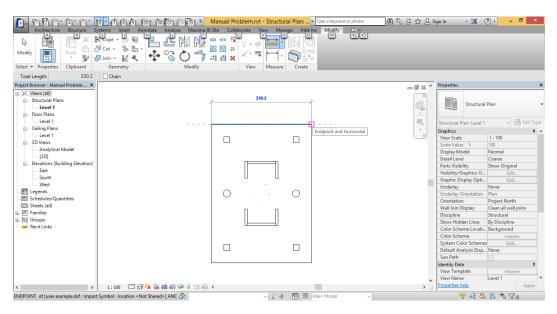


Figure (24): Scale the import .dxf file in Autodesk Revit

Then the user checks the location of the model in the positive quarter as shown in figure (25):

To move the model:

Modify → Move

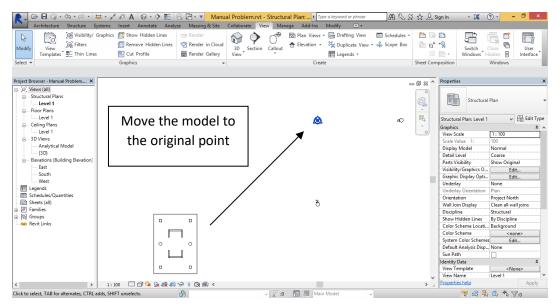


Figure (25): location of the import .dxf file in Autodesk Revit

The user will draw slab, beams, columns and cores above the imported AutoCAD similarly as in draw in Autodesk Revit.

Before explaining how the user can solve the lateral analysis using TBPAK.exe, there are general precaution that the user should put it into consideration before the analysis the lateral package:

• What is the difference between the fixed end beam and the released beam in Autodesk Revit?

The fixed end beam is the beam <u>fully or partially located above the column</u> as shown in figure (26)

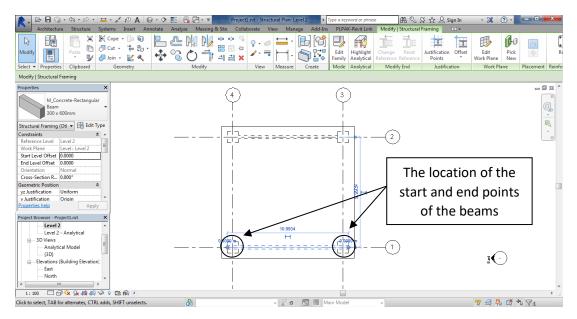


Figure (26): Simulation of fixed beam in Autodesk Revit

This beam will appear in .gen file of the lateral analysis as shown in figure (27) (the .gen file will explained later in solving lateral analysis using TBPAK.exe).

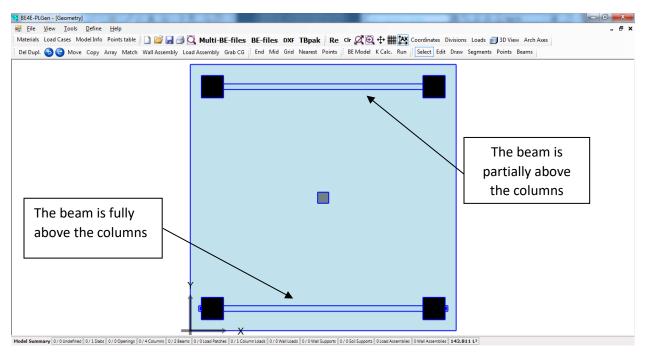


Figure (27): Simulation of fixed beam in PLGen.exe

While the released beam in Autodesk Revit is simulated as shown in figure (28), where the start and end point of the beam is **just linked to the vertical element** (column, shear wall, core).

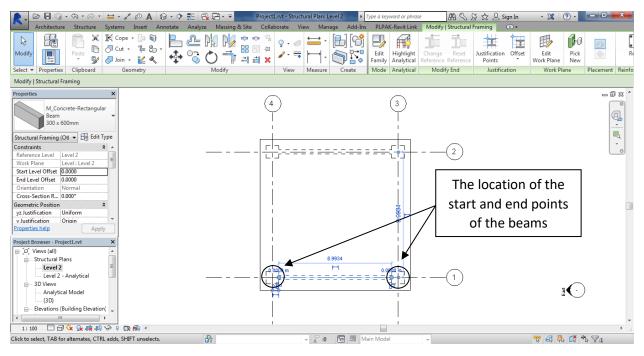


Figure (28): Simulation of released beam in Autodesk Revit

To move the start/end point for beams freely:

Select the beam — Right click on the start/end point — Disallow Joint

This beam will appear in .gen file of the lateral analysis as shown in figure (29) (the .gen file will explained later in solving lateral analysis using TBPAK.exe).

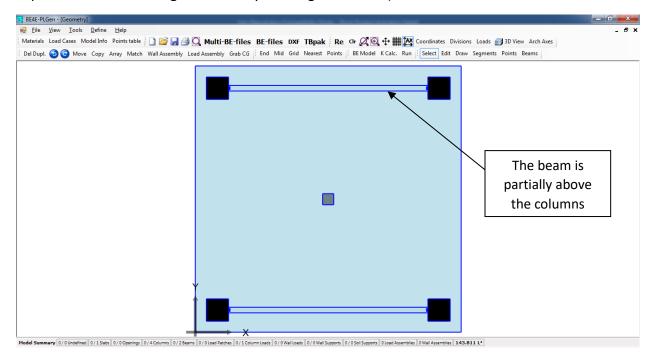


Figure (29): Simulation of released beam in PLGen.exe

In case of existing of beams/drops in the lateral model, the column's height need to be less than the depth of the beams, while the analytical lines for beams/drops and vertical elements should be linked together as shown in figure (30).

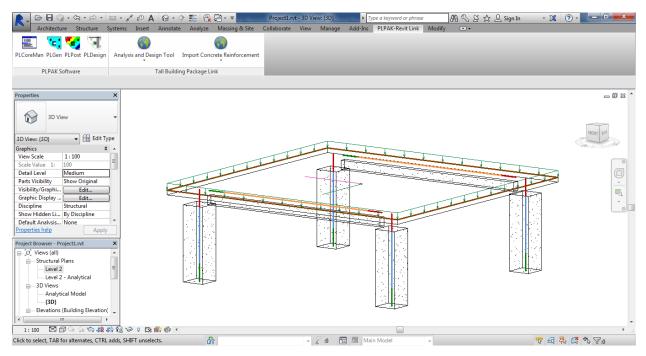


Figure (30): 3D view for beam column model

To change column height at the same time, keep the analytical line as it is: In 3d view select all model \longrightarrow filter \longrightarrow check none \longrightarrow check on structural columns \longrightarrow Detach top/bottom \longrightarrow Detach all \longrightarrow select each column \longrightarrow change top offset \longrightarrow select all model \longrightarrow filter \longrightarrow check none \longrightarrow check on analytical columns \longrightarrow in Properties window change top extension into projection and top x-projection into top level reference. • How can the user change the analytical boundary of the slab, as the analytical line is attracted automatically to the center of the columns as shown in figure (31)

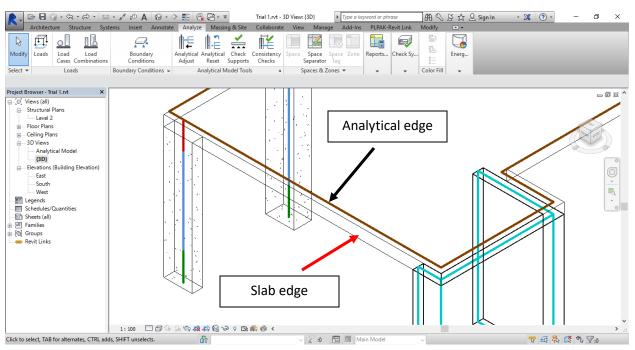


Figure (31): The location of slab edge and the analytical model.

To change the analytical edge to the slab edge:

Analyze menu → Analytical models tool → change all tolerance from 0.3 to 0.01 and remove all check box from Analysis/Physical Model Consistency Check.

• How can the user model drop in Autodesk Revit?

The user can insert drop in his model by adding structural slab with total thickness (slab thickness in addition to drop thickness) as shown in figure (32).

Please note the default drop discretization in .gen file is (4×4) the user can edit this division by opening the PLGen.exe and change it manually.

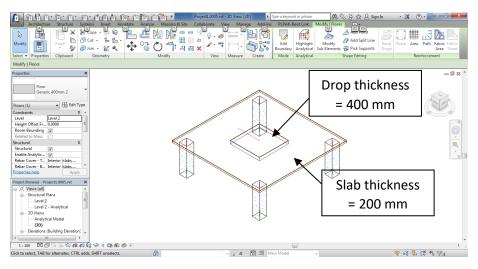


Figure (32): 3D view for slab drop model

B- Solving lateral analysis in TBPAK.exe:

After modeling the whole building in Autodesk Revit it is time to analyze using TBPAK.exe. From PLPAK-Revit Link as shown in figure (33), click on Analysis and Design Tool as shown in figure (34) finally choose tall building wizard to open the toolbox as shown in figure (35).

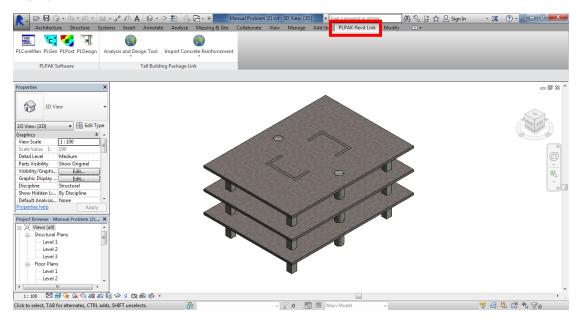


Figure (33): PLPAK-Revit Link

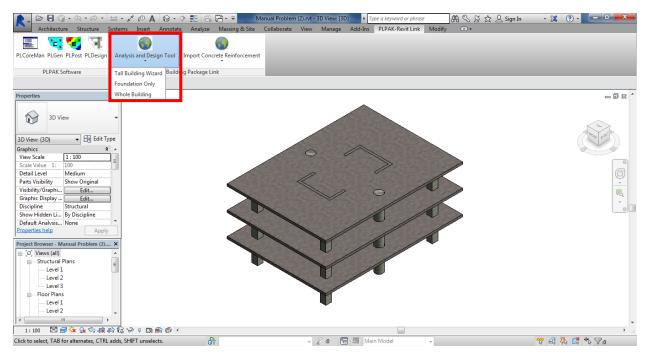


Figure (34): Analysis and Design Tool

💀 Tall Building Tool			
The following is the process and design of a tall building supports. Follow, the steps sequentially.	on fixed	Log of actions:	
1- Export text files for PLPAK. Export Textfiles 2- Convert and create all BEF)		
Create BEFILES You may choose eil step 3 (solve on fixed or step 4 (extract stiffned	her: base)		
3- Run the TBPAK to solve y	our structure.		
	ract Stiffness		
4- Postprocess with either P	POST or PLDesig	n. List of Floors: (Click to view in either F	LPOST or PLDesign)
PLPost	PLDesign		•
5- View in OpenGL viewer and design vertical elen OpenGL Viewer 6- Show reinforcement detail Return to Revit			

Figure (35): Tall Building Wizard

First press on Export Textfiles to open browse for folder to put the files in it, as shown in figure (36) then starting to exporting files as shown in figure (37), until all files are exported as shown in figure (38).

💀 Tall Building Tool		
The following is the process for and design of a tall building on supports. Follow, the steps belo sequentially.	fixed	
1- Export text files for PLPAK. Export Textfiles	Browse For Folder	
2- Convert and create all BEFILEs	Get Exported Files Directory	
Create BEFILES You may choose either step 3 (solve on fixed ba or step 4 (extract stiffness o		
3- Run the TBPAK to solve your s	B D B CUFEBE	
Solve on GPU (check to solv		
4- Postprocess with either PLPO		-
PLPost PLD	Recycle Bin	
5- View in OpenGL viewer and design vertical element	s 2 floors (4 columns)	
OpenGL Viewer	Make New Folder OK Cancel	
6- Show reinforcement detailing.		
Return to Revit		

Figure (36): Export text files

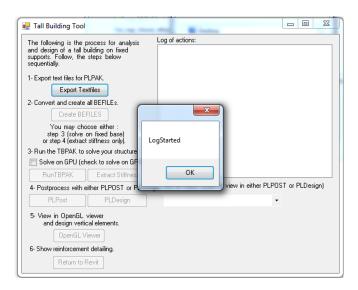


Figure (37): Start export text files

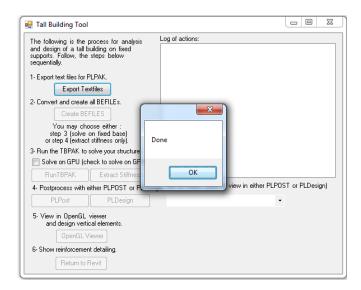


Figure (38): finish export text files

Second press on Create BEFILES as shown in figure (39), then finishing these files by opening the PLGen.exe as shown in figure (40)

💀 Tall Building Tool	
The following is the process for analysis and design of a tall building on fixed supports. Follow, the steps below sequentially.	Log of actions: Structural components and loading have been exported. Boundary element files have been created for each floor.
1- Export text files for PLPAK, Export Textfiles 2- Convert and create all BEFILEs. Create BEFILES You may choose either : atep 3 (cohe on fixed base) or step 4 (extract stiffness only). 3- Run the TBPAK to solve your structure.	
Solve on GPU (check to solve on GPU) RunTBPAK Extract Stiffness	
4- Postprocess with either PLPOST or PLDesig	n. List of Floors: (Click to view in either PLPOST or PLDesign)
PLPost PLDesign	•
5- View in OpenGL viewer and design vertical elements. OpenGL Viewer 6- Show reinforcement detailing. Return to Revit	

Figure (39): Create BEFILES

BE4E-PU	Gen -	(Geom	etry																																		
File	View	Too	ls I	Define	н	elp																															- 8
laterials	Lord	Cases	Mod	el Info	Poi	ints ta	ble	🗳 🖥	14	D	Mu	lti-E	3E-fil	les	BE-	files	DX	F TB	pak	R	e Cle	Ø	Ð, 4	ŀ#	1AX	Co	ordinate	s D	ivision	s Loa	ds 💼	30	View /	Arch Au	es		
Del Dupl.																																					
													The back	Multi-	-BENI eir pa	e conv	erter	has cot	nplete	ed 3 fic	oors co	rrectly	and		ed the	m											
									Y																												

Figure (40): Finishing create BEFILES

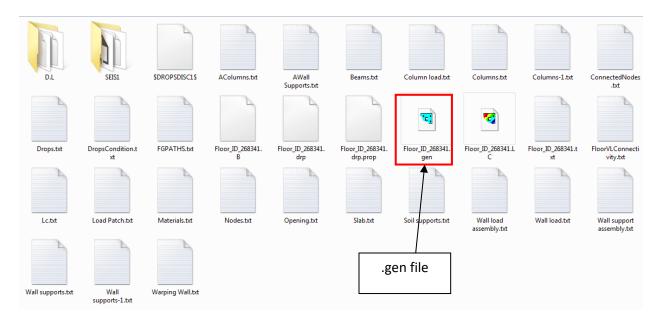
From this step the user should check his model as shown from figure (41) to figure (43) at each floor and the user can also change anything in the model before solving the whole building for example change the boundary element for the slabs or change number of division for beams, shear wall, cores, drops...etc.

To check the model:

Click on the folder where the analysis located \longrightarrow open each floor ID \longrightarrow open .gen file for each floor to check your model.

퉬 Floor_ID_268341	12/5/2017 11:47 PM	File folder	The three
퉬 Floor_ID_271959	12/5/2017 11:47 PM	File folder	
Floor_ID_272002	12/5/2017 11:47 PM	File folder	floor levels ID
퉬 Lateral System Files	12/5/2017 11:46 PM	File folder	
SMat\$	12/5/2017 11:46 PM	File	1 KB
SHEARAREA\$	12/5/2017 11:46 PM	File	1 KB
ASM.txt	12/5/2017 11:46 PM	Text Document	5 KB
DF.D.L	12/5/2017 11:46 PM	L File	1 KB
DF.SEIS1	12/5/2017 11:46 PM	SEIS1 File	1 KB
FloorsDirectories.txt	12/5/2017 11:46 PM	Text Document	1 KB
GenDirectories.txt	12/5/2017 11:46 PM	Text Document	1 KB
📋 GenNames.txt	12/5/2017 11:46 PM	Text Document	1 KB
LCLIST	12/5/2017 11:46 PM	File	1 KB
📔 Log.txt	12/5/2017 11:46 PM	Text Document	100 KB
📔 NodalForce.txt	12/5/2017 11:46 PM	Text Document	1 KB
StructureVLConnectivity.txt	12/5/2017 11:46 PM	Text Document	1 KB

Figure (41): Floor levels ID





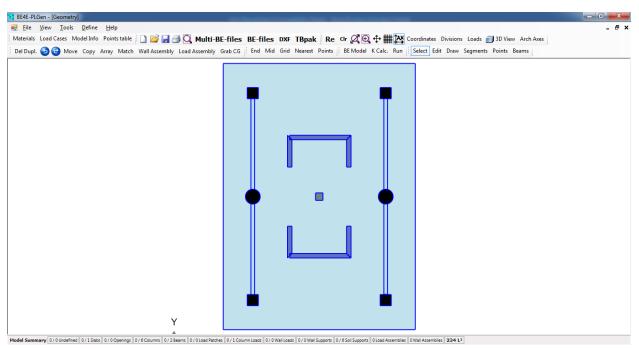


Figure (43): Boundary element model for each floor level

Once the .gen files are accepted for the user, the user return to the tall building wizard to run the TBPAK.exe as a **third step** as shown in figure (44).

In case of existing of frame resistance in building model, the wall control window will open and the user will click on start and next only.

While in case of existing of cores or shear walls in building model, the wall control will open and the user will click on start and right on the left hand side the corner nodes for every core or shear wall anti clockwise as shown from figure (45) to (47).

🖳 Tall Building Tool	
The following is the process for analysis and design of a tall building on fixed supports. Follow, the steps below sequentially.	Log of actions: Structural components and loading have been exported. Boundary element files have been created for each floor.
1- Export text files for PLPAK. Export Textfiles	
2 Convert and create all BEFILEs. Create BEFILES You may choose either : step 3 (solve on fixed base)	
or step 4 (extract stiffness only). 3- Run the TBPAK to solve your structure. Solve on GPU (check to solve on GPU)	
RunTBPAK Extract Stiffness 4- Postprocess with either PLPOST or PLDesign	_{n.} List of Floors: (Click to view in either PLPOST or PLDesign)
PLPost PLDesign S- View in OpenGL viewer and design vertical elements. OpenGL Viewer G- Show reinforcement detailing. Return to Revit	•

Figure (44): Run TBPAK.exe

Wall control	vall corners in anticlock	wise direction:			
	Point number			REFRESH	
					*
		Spacir - maxir	ng (boundary discretization num 0.01):	0.01	
			START	NEXT	

Figure (45): Wall control window

	Point number		REFRESH		
	8			1 4.3,2.15	
	7			2 4.3,0	
	2			3 4,0.3000002	
	1				
	4			4 4,2.15	
	3			5 0.3000001,2.15	
	6	85	41	6 0.3000001,0.3	
•	5			7 0,1.144409E-08	
*				8 0,2.15	
				9 4.3,0	
		710	32	10 0.1500001,0	
		Spacing (boundary discretiza - maximum 0.01):	ation 0.01		
		START	NEXT		



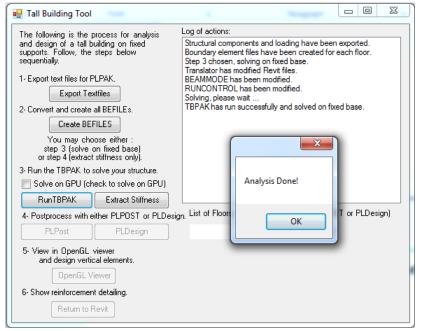


Figure (47): Finishing TBPAK.exe

C- Showing straining actions in PLPost.exe:

Once the user finish analyzing in TBPAK.exe, he should choose the floor ID to show the straining action on it as shown in figure (48) and figure (49).

Tall Building Tool The following is the p and design of a tall bu supports. Follow, the sequentially. Texport text files for P Export Tex Create BEF You may chou step 3 (solve c or step 4 (extract 3-Run the TBPAK to s Solve on GPU (ch BunTBPAK	ilding on fixed steps below titiles all BEFILEs. ILES Dase either : in fixed base] stiffness only).	Log of actions: Structural components and loading have been exported. Bounday element files have been created for each floor. Step 3 chosen, solving on fixed base. Translator has modified Revit files. BEAMMODE has been modified. RUNCONTROL has been modified. Solving, please wait TBPAK has run successfully and solved on fixed base.
4- Postprocess with ei	ther PLPOST or PLDes	ign. List of Floors: (Click to view in either PLPOST or PLDesign)
PLPost	PLDesign	Floor_ID_268341\Floor_ID_268341
5- View in OpenGL v and design vertic OpenGL Vi 6- Show reinforcement Return to R	al elements. ewer : detailing.	

Figure (48): Open PLPost.exe

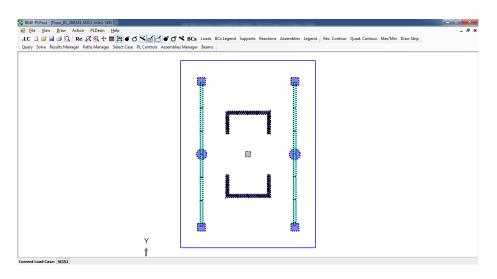


Figure (49): Open PLPost.exe

First from the bottom lift of the screen the user can press double click on current load case to open load combination window as shown in figure (50).

Second there is an icon called result manager shown in figure (51), where the user can view three types for slabs straining actions as shown from figure (52) to (54):

- 1- Strip straining action
- 2- Overall contour
- 3- Certain quadratic/rectangular local contour

for more information about the PLPost.exe, Please read user manual for PLPost

Select Case		x
SEIS1 D.L Load Combination	Combination Name: Load Comb	ination
	Add Entry Remove Entry	 1.12 1.28 0
ADD REMOVE		OK Cancel

Figure (50): Load combination window

Results Manager	
Strips ID: Enabled: Export Npoints: Result: • Statt: End: Status: • theta:	Strip Staining action manage
Contours Main Contour Enabled: Export N: 8 Spacing: 0.5	
Min Is Userdefined: min: 0 Max Is Userdefined: max: 0 Status: ToBeSolved Current Variable: Mox V	Overall contour manage or quadratic contour manage
Image: Column Plot Enabled: N: 8 Max Is Userdefined: max: 0 Current Variable: FZ V Min Is Userdefined: min: 0 Refresh Show Total Forces: Export Close Close	

Figure (51): Result manger window

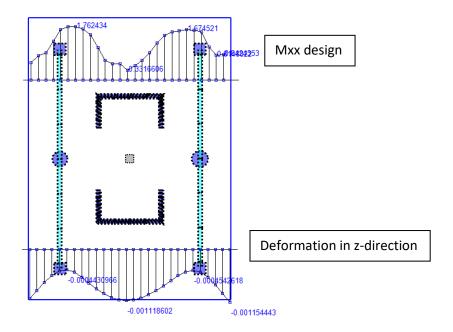


Figure (52): Strip Straining action

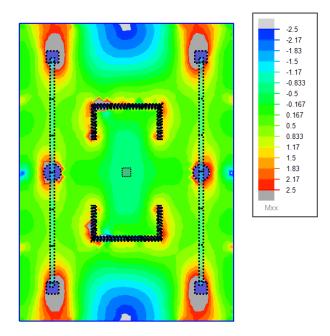
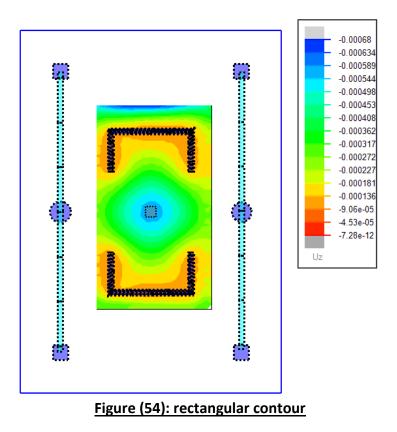


Figure (53): overall contour



Third the user can read the beam straining action from icon beam as shown in figure (55).

BeamsManager		×
Show Nodes:	Show Nodes Nos:	Read Beam Geometry
Show Directions:		Read Default Beam Results
Show Result: Show Result Value:		Browse Required Beam Results
0		

Figure (55): beams manager window

once the user check box on show results and show results value, then click on read beam geometry and read default beam results. The beams straining action appear according to the type of straining action need to show as shown in figure (56).

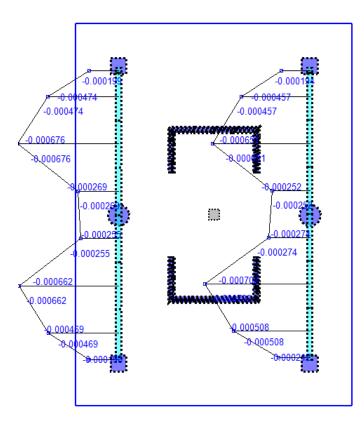


Figure (56): beams straining actions

Fourth, the user can show the columns straining actions by two steps:

- 1- Export from the PLGen the assembly file for the solved floor as shown in figure (57).
- 2- Load the assembly file in PLPost as shown in figure (58).

 B 	E4E-P	LGen - [Geo	metry]																				
	File	View T	ools Define	Hel	р																		- e ×
M		New .gen	Ctrl+N	Poin	ts table	1 🗋 📂	H 6	3 Q.	Multi-E	E-files	BE-fi	les D	oF TBpa	k Re	ch ؀	€⊕∰⊉	Coordina	ates Divi	isions Lo	oads 🗐	3D View	Arch Axes	
D	2	Open .gen	Ctrl+O													K Calc. Ru							
		Import	,	E.			-		-	_							_	-					
		Export	•		BE files																		
	-				Assemi	olies																	
		Save .gen			Beam a	ssemblies																	
	ð	Print	Ctrl+P		Text for	mat					T					T							
	<u> </u>	Print Previe	w	—																			
		Page Setup		I .																			
		Exit	Alt+F4	1																			
	-			1																			
													-										
											T.			- ·		—							
													и <u> </u>										
																_ <u></u>							
																_							
								,															
							Y																
				10.00			1	1.		1			. 11			Tec							

Figure (57): export assembly file

Assembly Manager					
Add					
1 2	X ID:	1		Export	
2 3 4	Area:	0.5625	Theta:	0	
5 6	CGIs User Defin	ed: 🔽		Use Principal	
	CGOrdered Pair:	5.341,4.542			
	Inertia X:	0.02637635649695	Inertia XL:	0.02637635649695	
	Inertia XY:	1.07814495891034	Inertia XYL:	1.07814495891034	
Assemblycells:	Inertia Y:	0.02637986496688	Inertia YL:	0.02637986496688	
1.	Fz:	27.0707645	Fz:	27.0707645	
	Mx:	-4.52480173	Mx L:	-4.52480173	
	My:	2.510772	My L:	2.510772	
	Load	Export All			ОК
		-			

Figure (58): load assembly file

Then the user open result manager and check box on column plot enable and show total forces as shown in figure (59) and figure (60).

X	ID:	Enabled		Export
	Npoints:	Result:		
	Start:	End:		
		theta:		Use Local
	Status:	▼ theta.		
0		409		<u> </u>
Contours				
Main Contour X	ID: Main Con	tour Enabl	ed: 🔲	Export
	N: 16	Spaci	ng: 0.5	
	Min Is Userdefin	ed: 🔽 min:	-2.5	
	Max Is Userdefin	ed: 🔽 max:	2.5	1/x= -6.6
	Status: Solved	Currer	nt Variable:	Mox 🔻
	Itheta: 0			
olumn Plot	inieta. U			
			1151.01	r .
Enabled: 🔽 N: 8	Max Is Use	rdefined: 🔲 ma	ex: 1151.01	5
Current Variable: FZ	Min Is Use	rdefined: 🔲 m	in: -1234.3	
	Show Total	Forces:		Export
Refresh	Show Total	roices.		Export

Figure (59): columns result manager

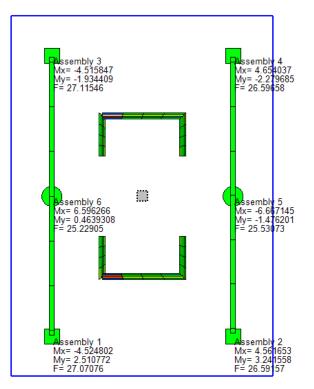


Figure (60): columns straining action

Getting Help

The BE4E.com customer support team is always welcoming problems and suggestions of registered customers. Just send an e-mail including your questions, or your model together with your questions to: plpak@be4e.com

Also check our site news at <u>www.plpak.com</u> regularly for *Problems and Solutions* section and the *Frequently Asked Questions* section