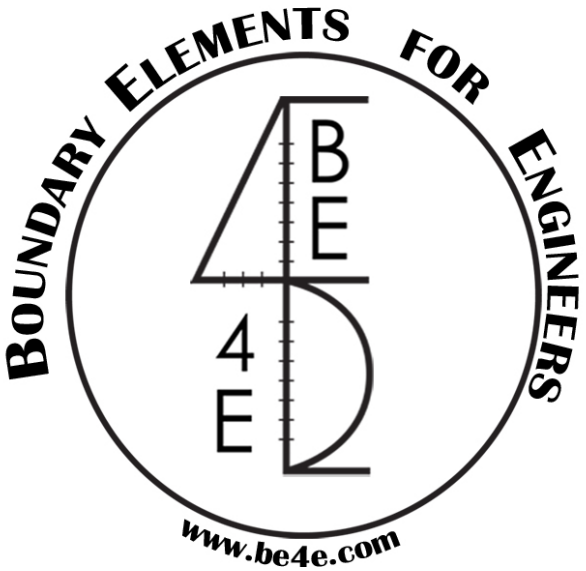


The Fixed Base Package

User MANUAL

PLPAK™ Version 2.00

STRUCTURAL ANALYSIS SOFTWARE USING

THE BOUNDARY ELEMENTS METHOD

Copyright © 2000-2022 https://www.plpak.com

E-mail: support@plpak.com

Disclaimer

Considerable time, effort and expense have gone into the development and documentation of the PLPAK™ software. The PLPAK™ software has been thoroughly tested and used. The PLPAK™ 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 PLPAK™ software. The user must explicitly understand the assumptions of the PLPAK™ software and must independently verify the results produced by the PLPAK™ software.

Copyright

Copyright © BE4E.com, 2000-2022
All rights reserved.

The PLPAK™, PLGen™, PLView™, PLCoreMan™, PLPost™, PL.exe™, PLDesign™, TBPAK™ are registered trademarks of BE4E.com..

The computer program PLPAK™ 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:

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

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

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:

First, 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.

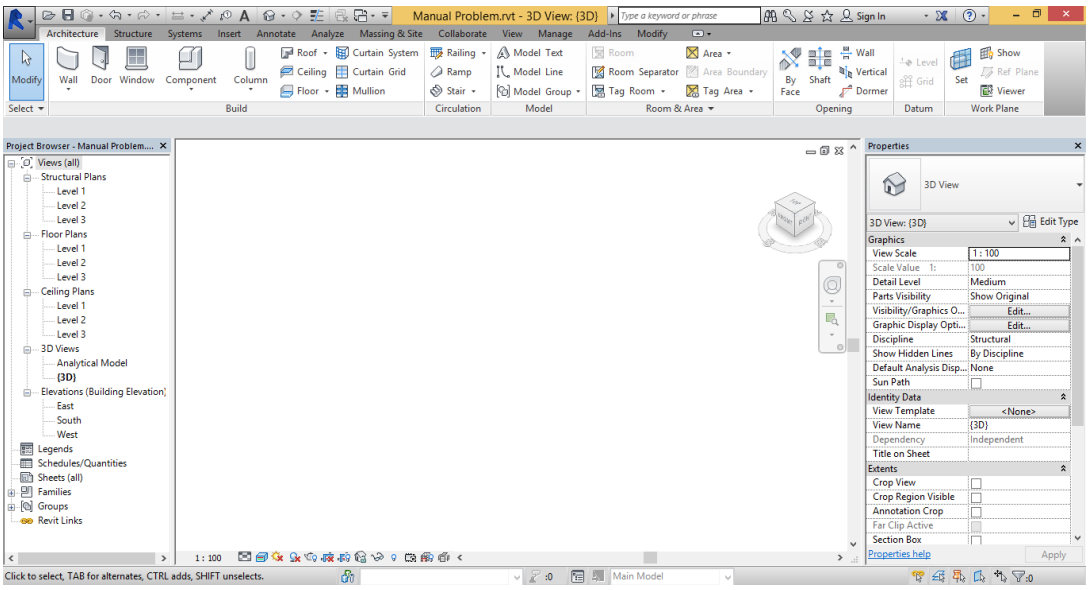


Figure (1): Structural Analysis-DefaultMetric-Up.rte file

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:
C:\ ⟶ Program files ⟶ PLPAK ⟶ at any place press right click to open the properties of the file ⟶ Check on the allow for all icons. (As shown in figure (2))

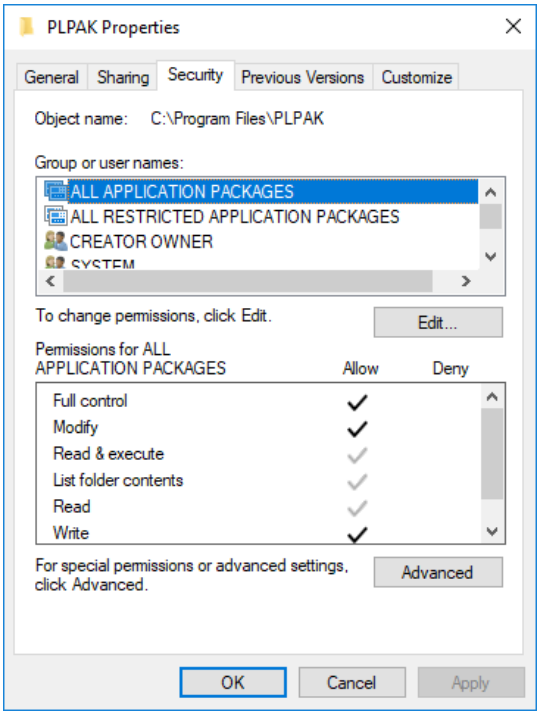
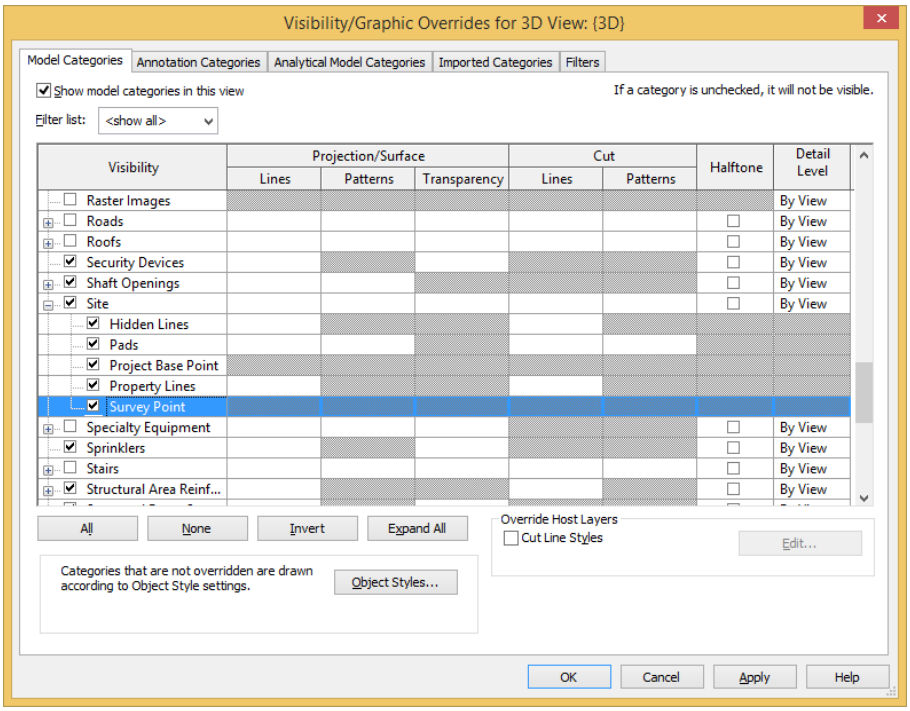


Figure (2): PLPAK Properties folder

Third, 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



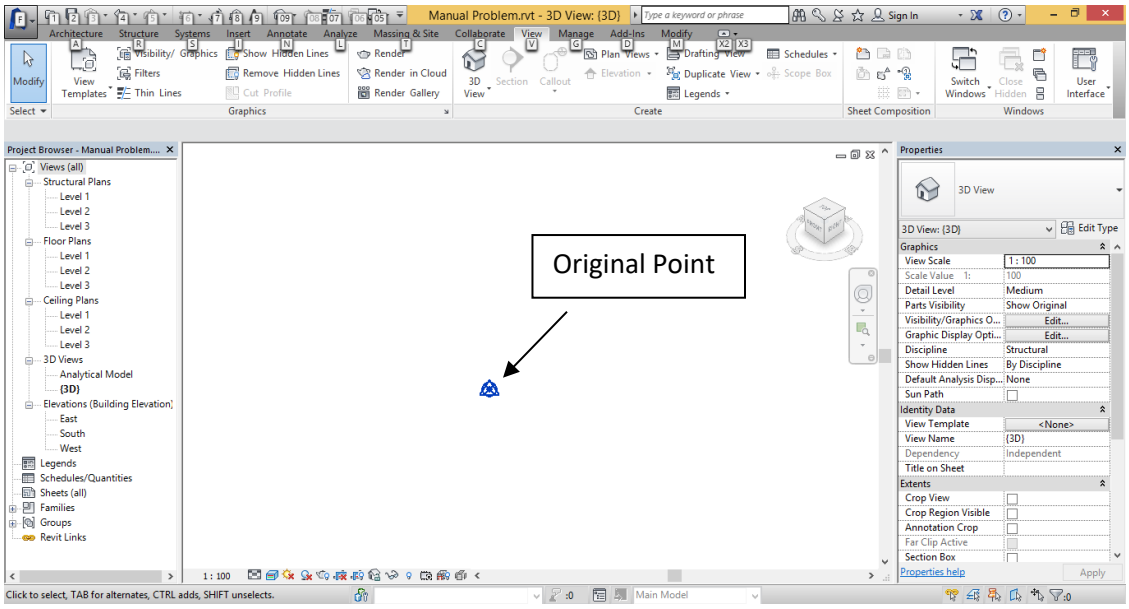


Figure (3): Show the original point in Autodesk Revit

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

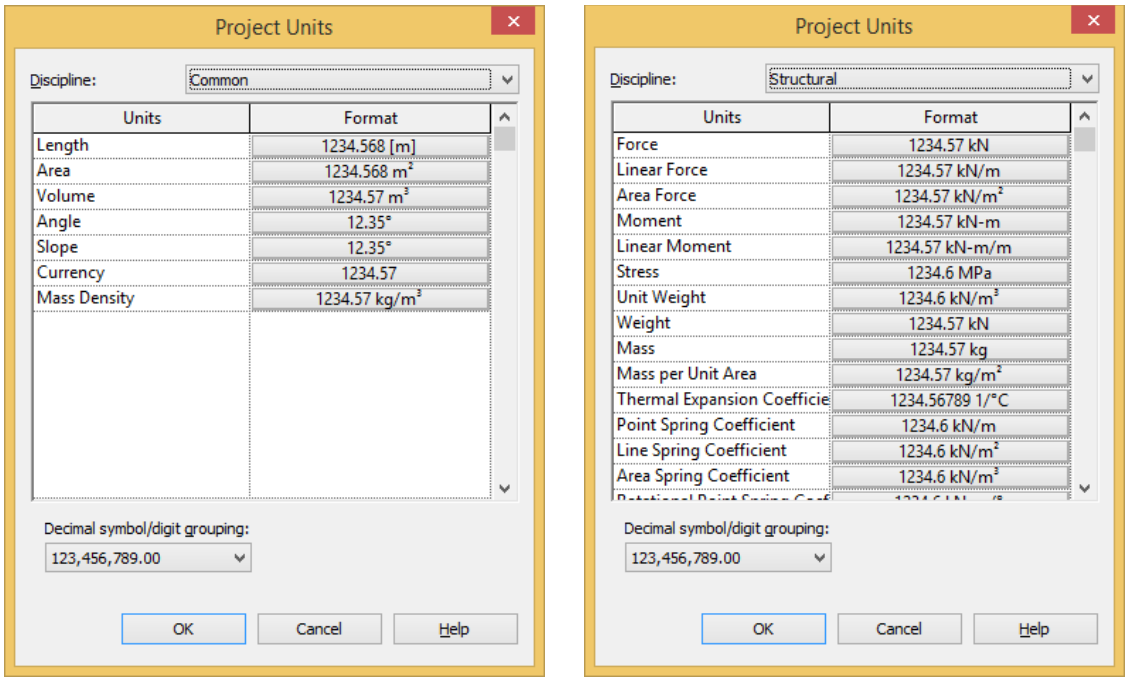


Figure (4): Project Units window

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)

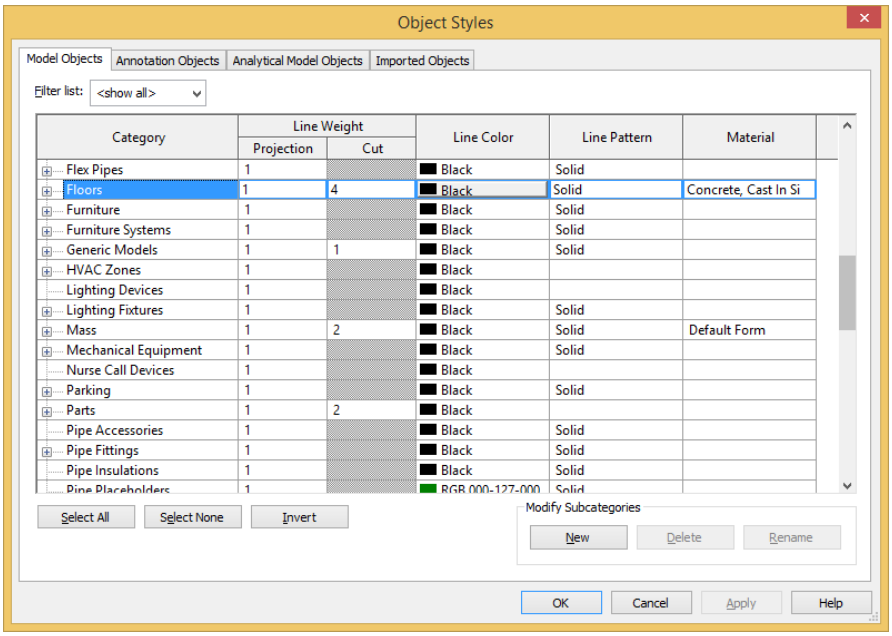


Figure (5): choose the material properties for slabs, beams and columns

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).

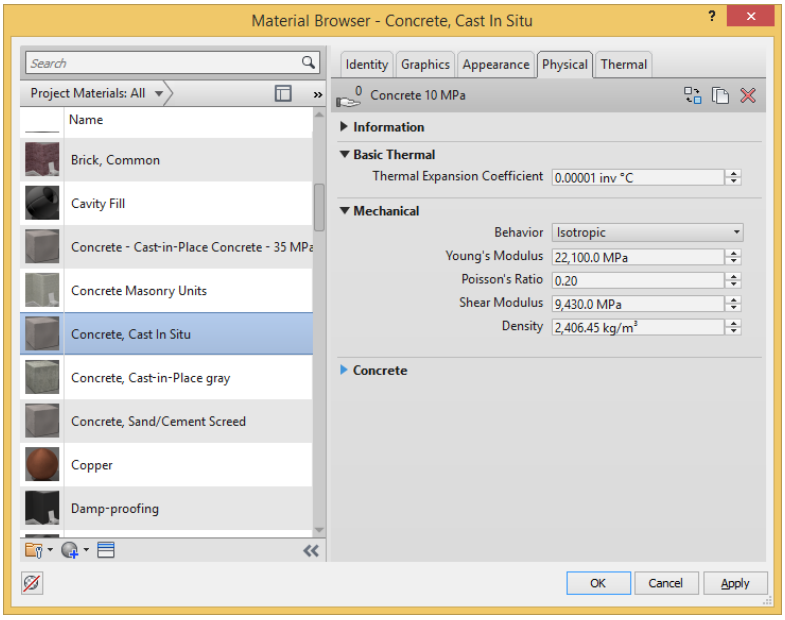


Figure (6): Material Browser window

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)

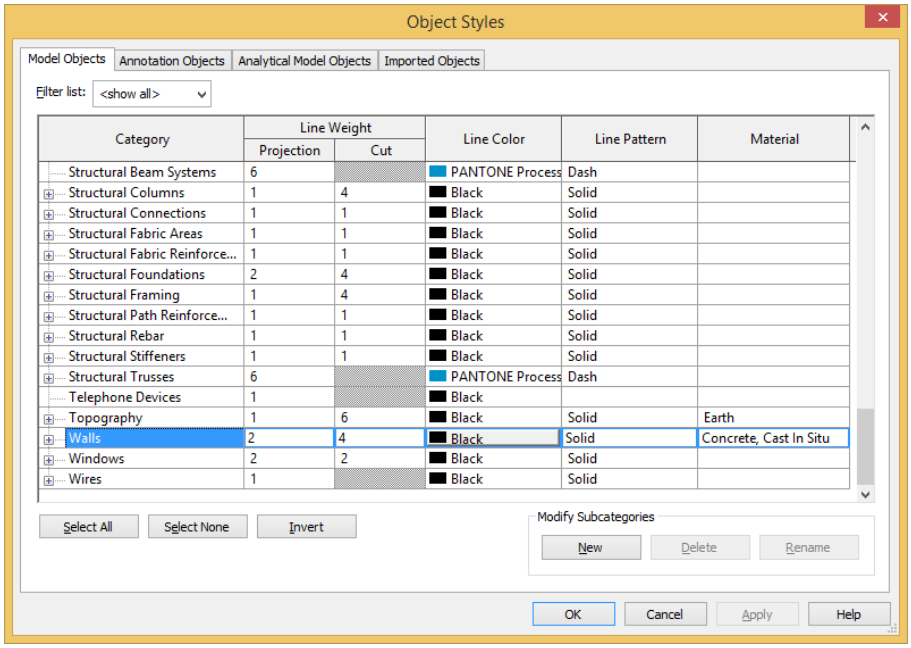


Figure (7): choose the material properties for walls

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).

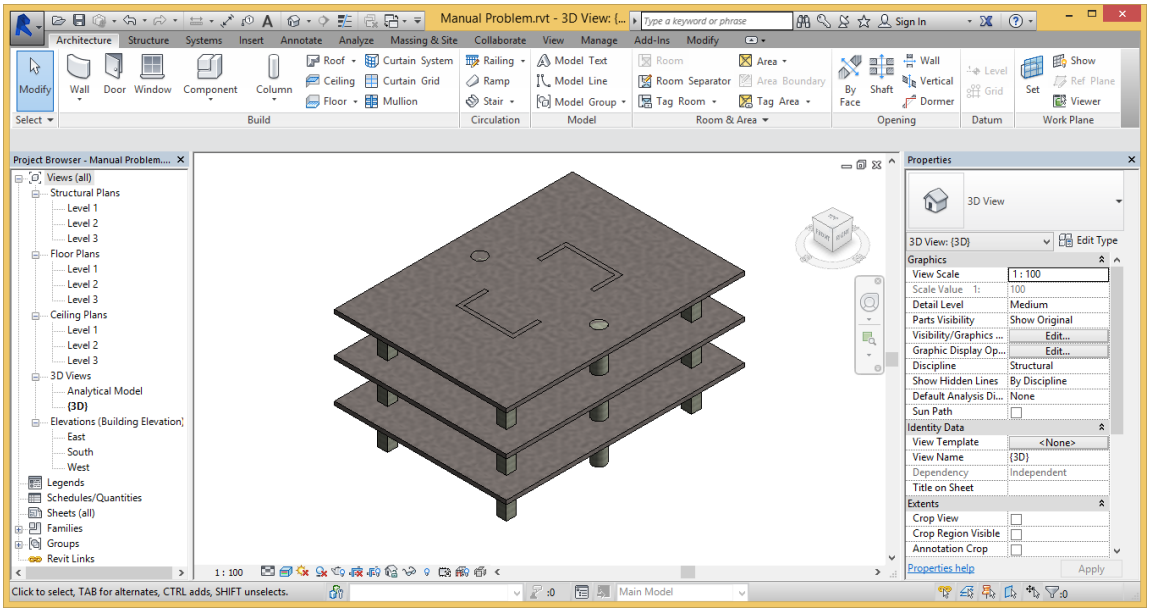


Figure (8): Autodesk Revit structural model

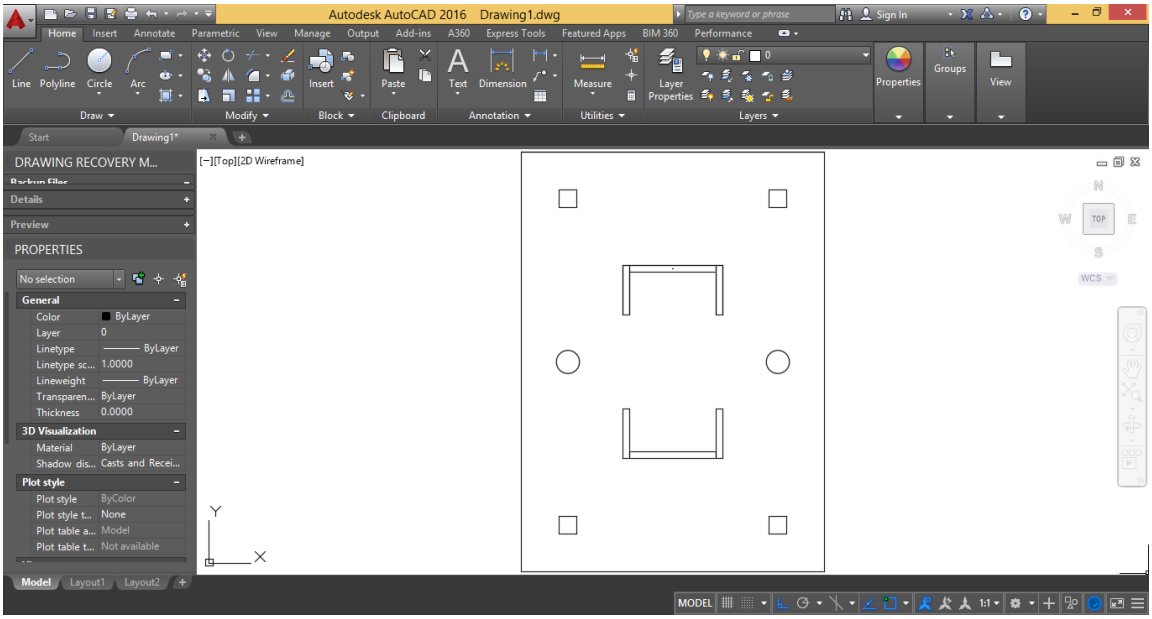


Figure (9): AutoCAD .dxf file

I- Draw from Autodesk Revit:

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

Structure ⟶ Grid

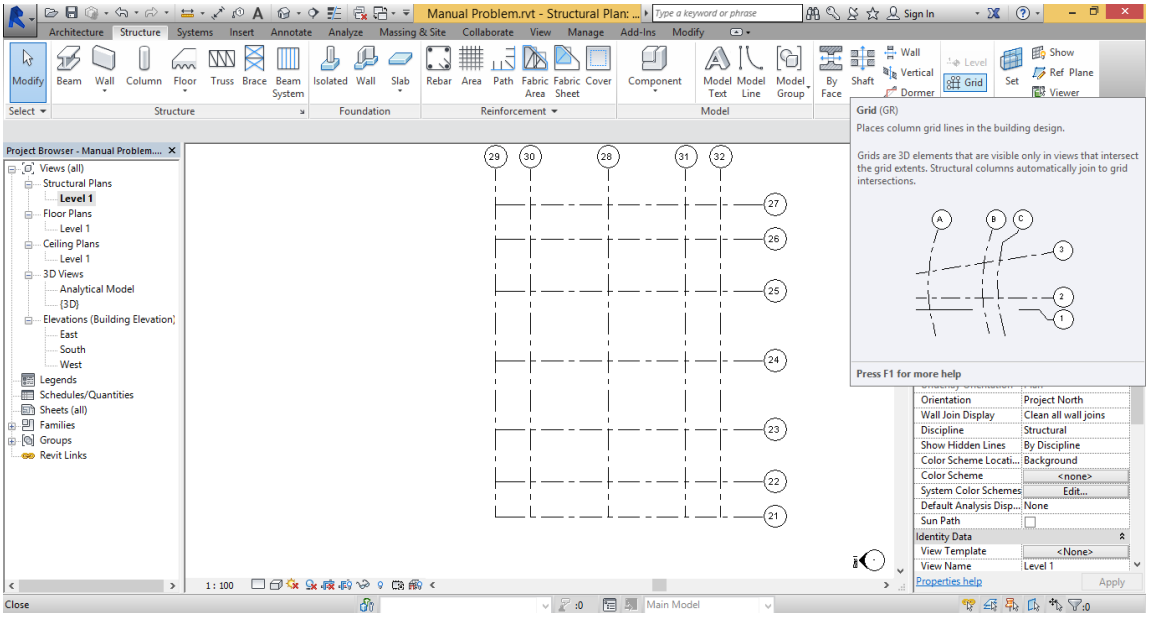


Figure (10): Grid Drawing

2- Draw the slab with the required properties as shown in figure (11)

Structure ⟶ Floor ⟶ Floor Structural

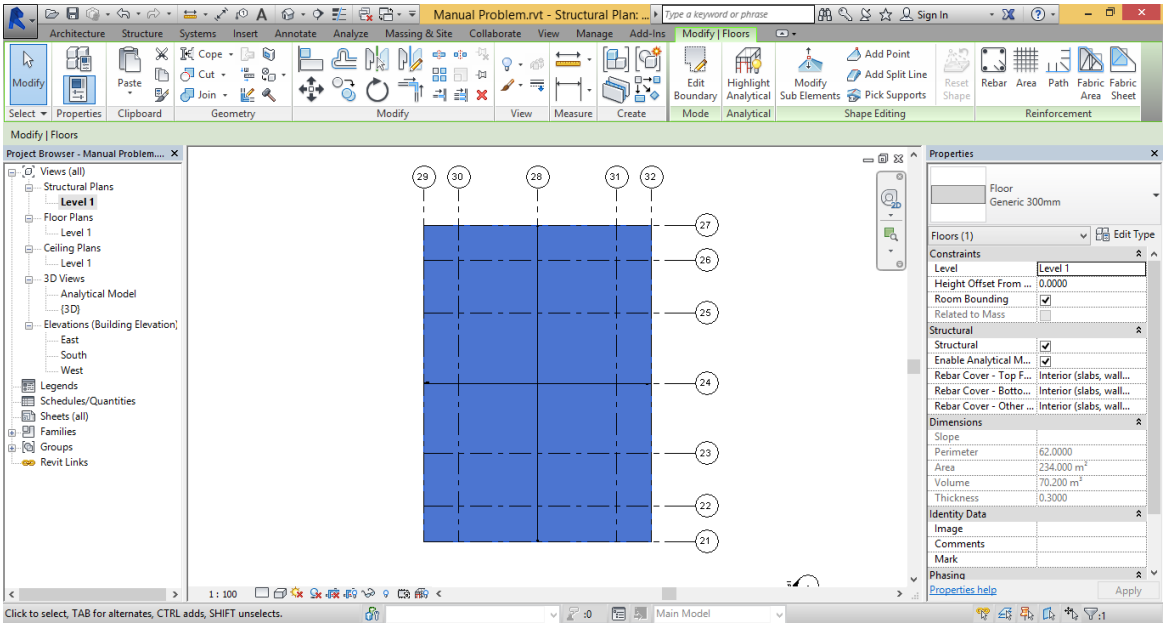


Figure (11): Slab Drawing

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).

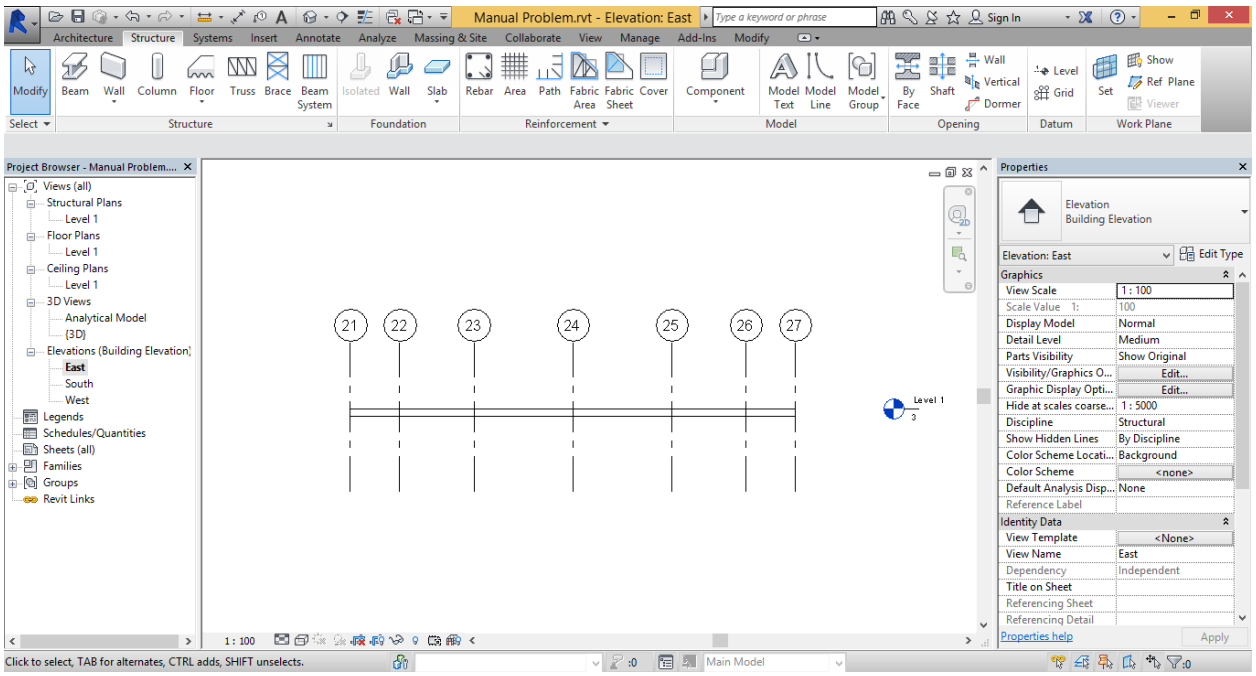


Figure (12): Check zero level

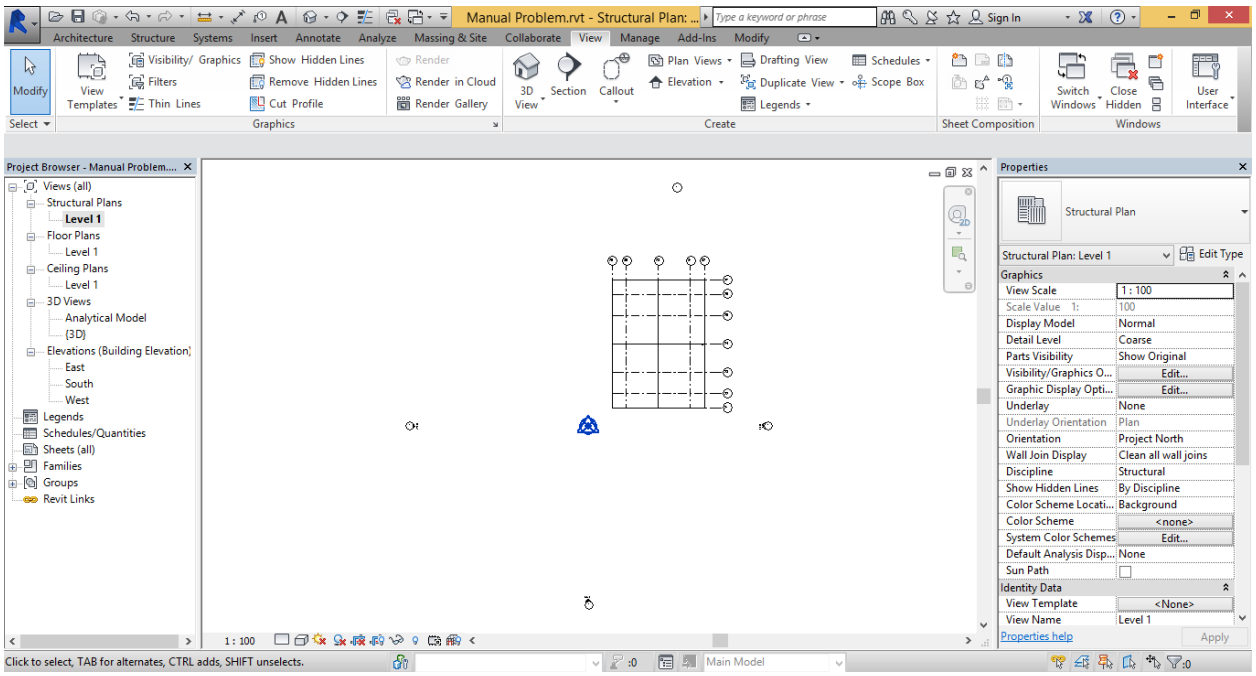


Figure (13): Check the model is in positive quarter

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

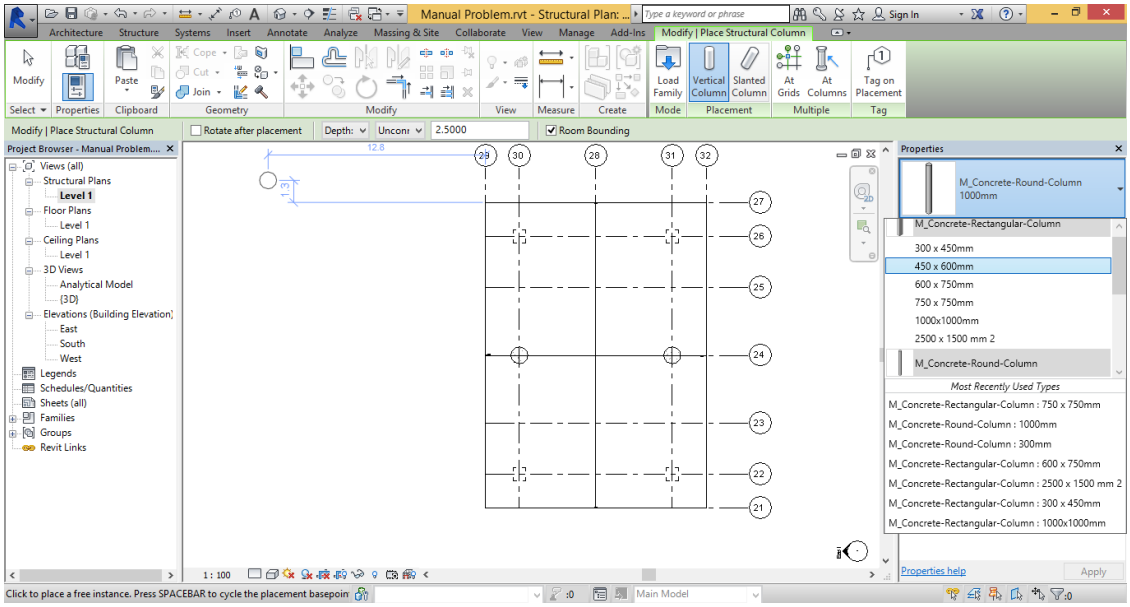


Figure (14): Columns drawing

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

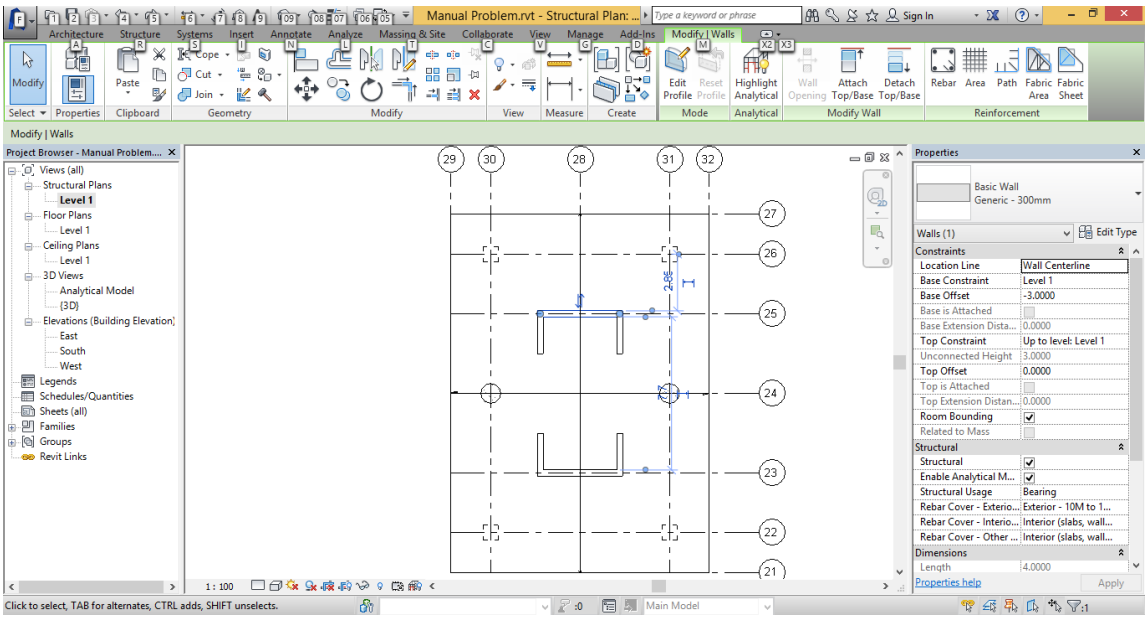


Figure (15): Cores and shear walls drawing

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

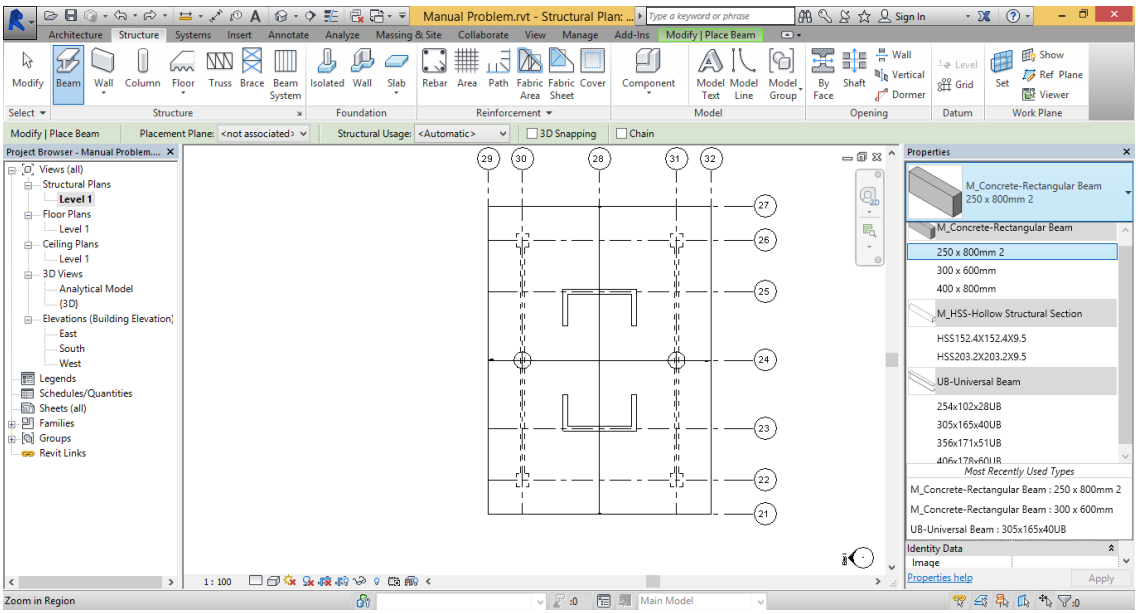


Figure (16): Beams drawing

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

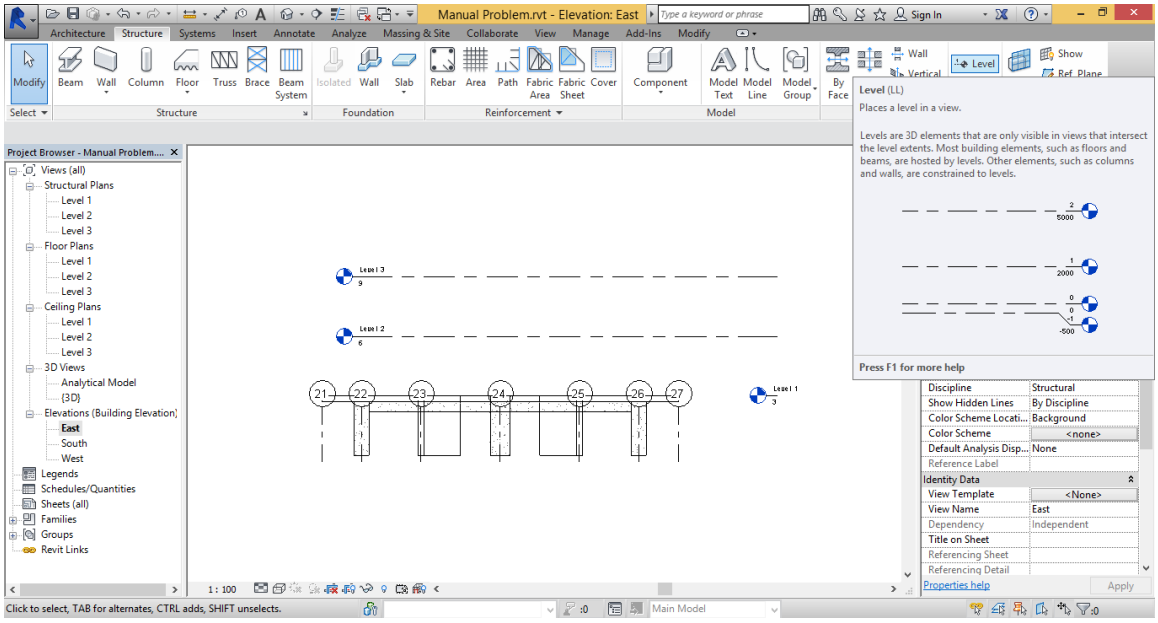


Figure (17): Determine levels

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

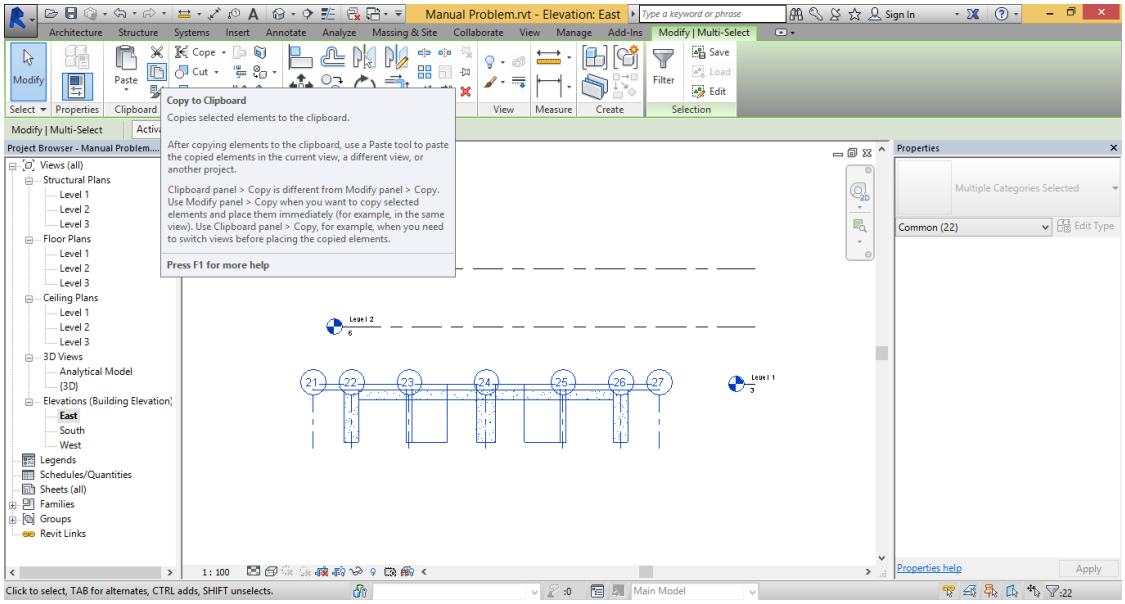


Figure (18): Select and Copy drawn level

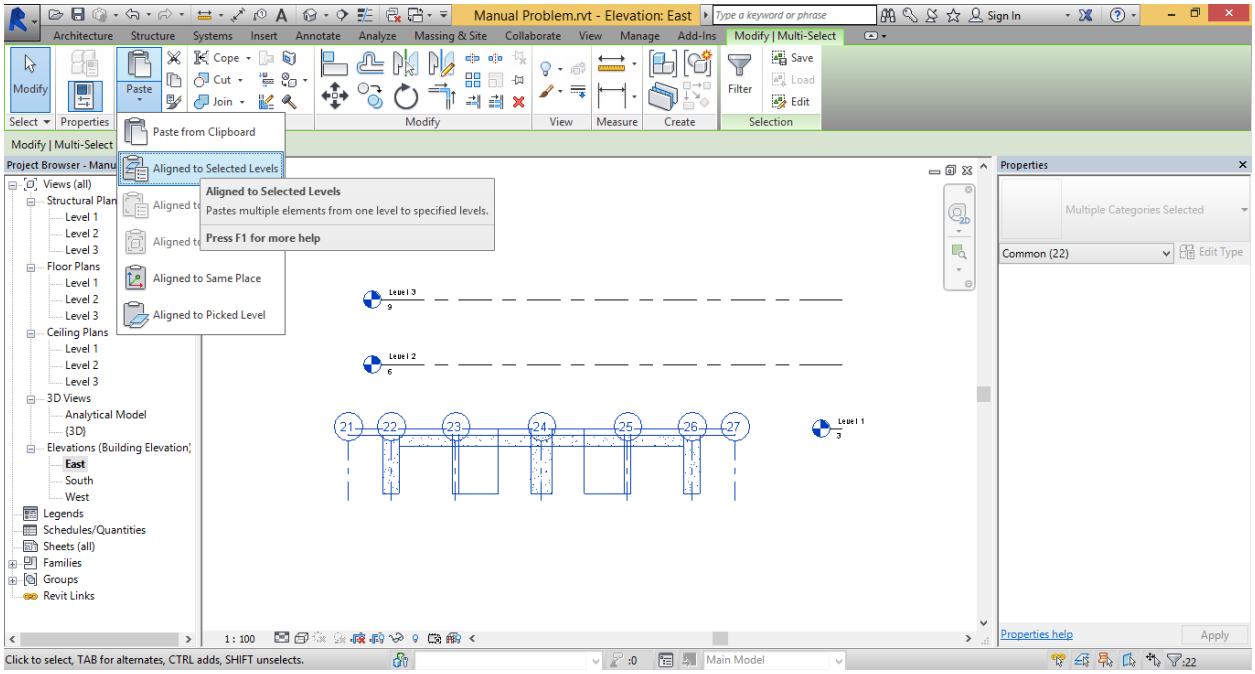


Figure (19): Paste the copied level on the selected levels

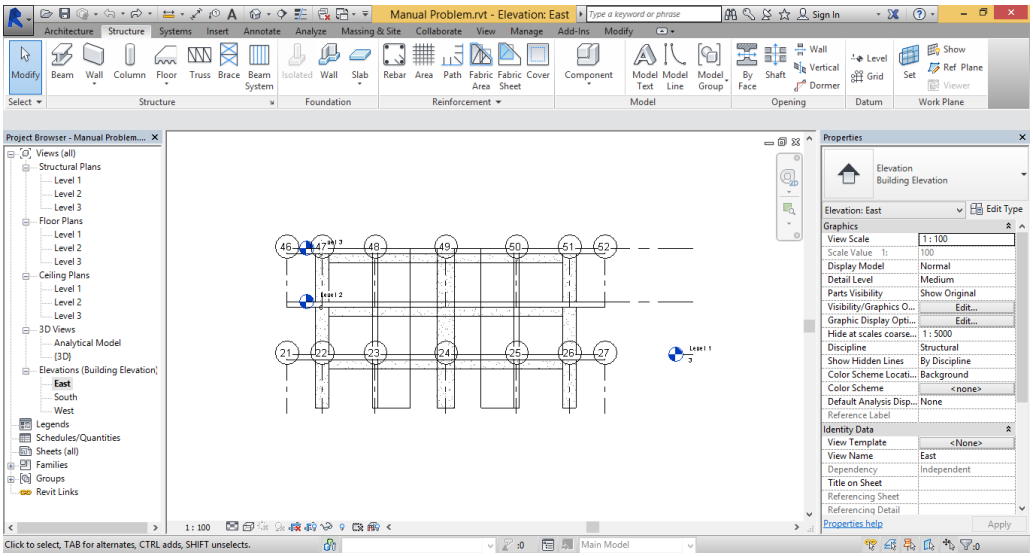


Figure (20): Modeling the whole building

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

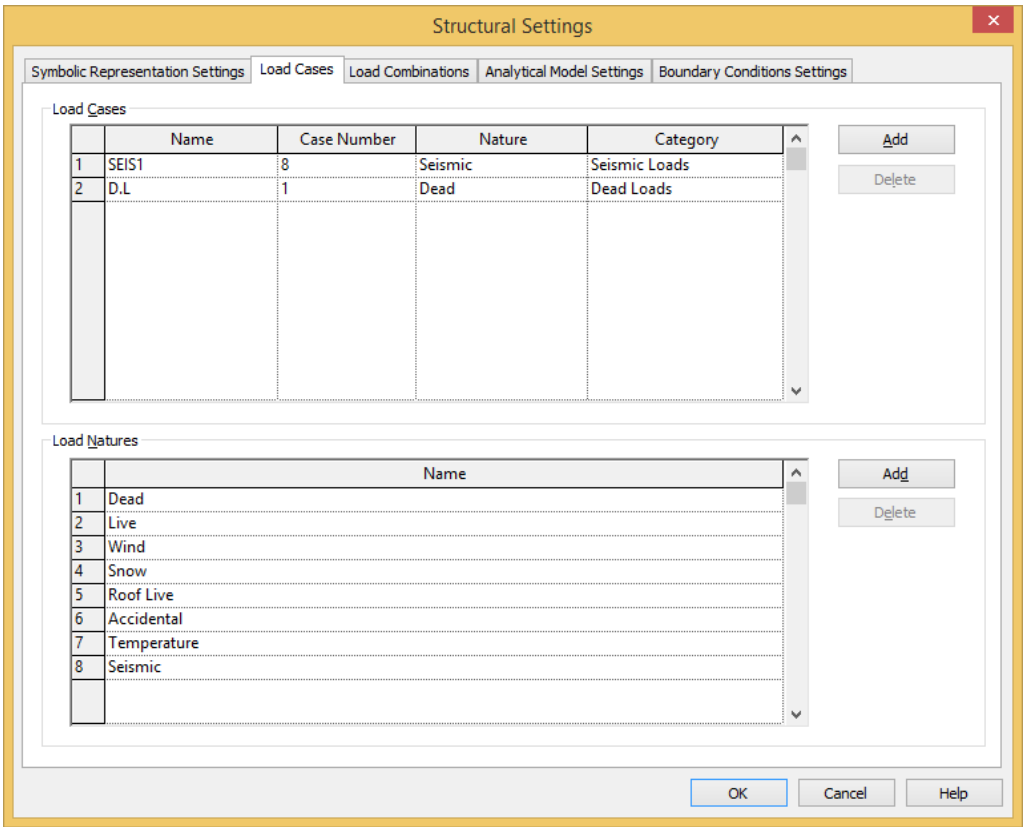


Figure (21): Load Cases determination

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

Analyze → Loads → Point Load
Loads → Line Load
Loads → Area Load

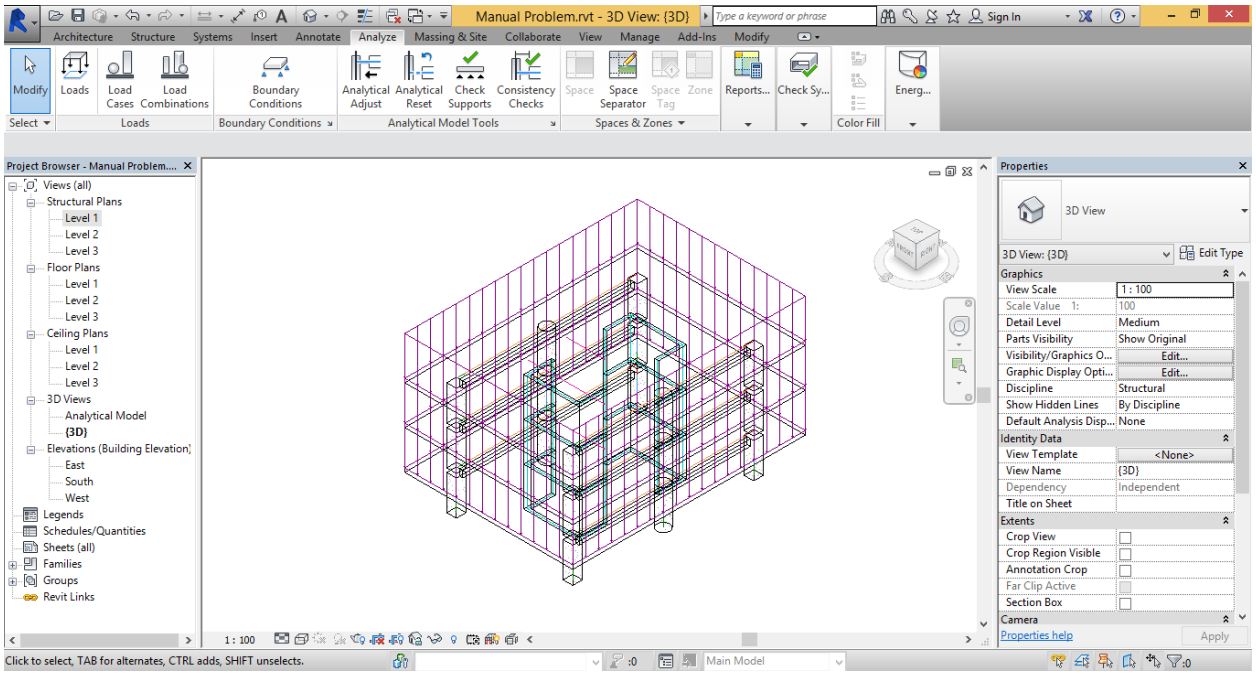


Figure (22): Loads in different load cases

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

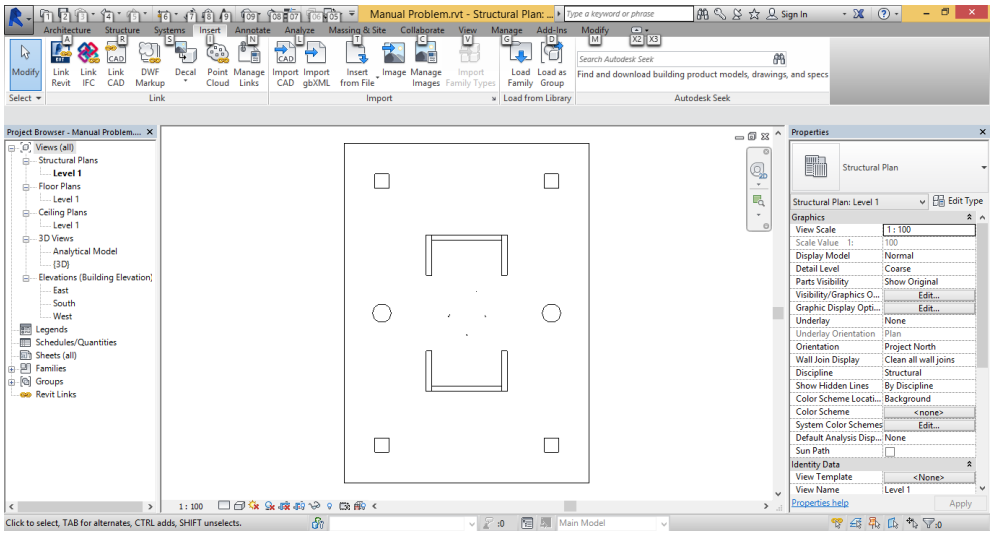


Figure (23): Import .dxf file in Autodesk Revit

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

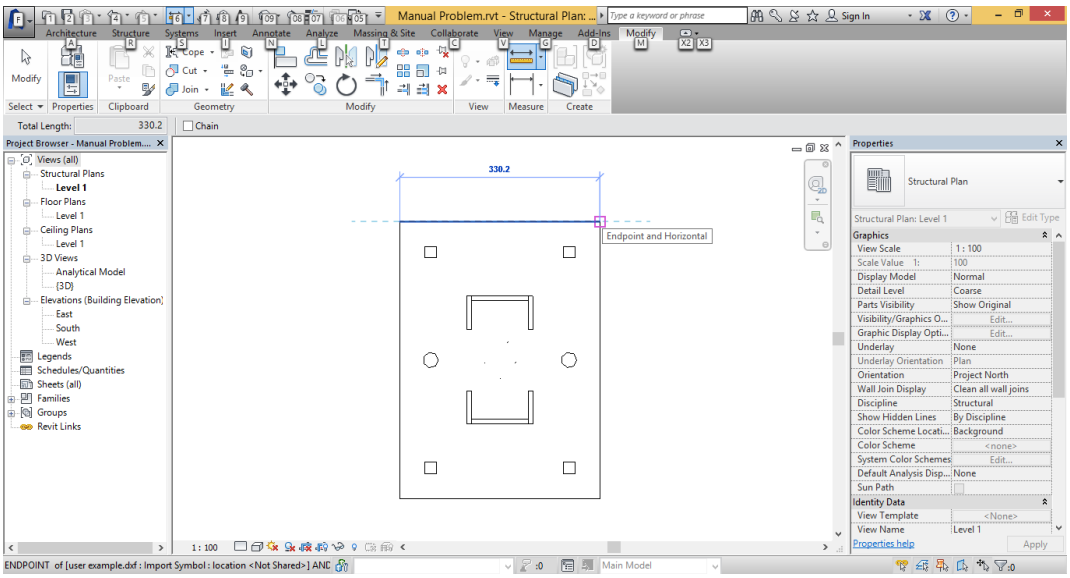


Figure (24): Scale the import .dxf file in Autodesk Revit

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

To move the model:
Modify ⟶ Move

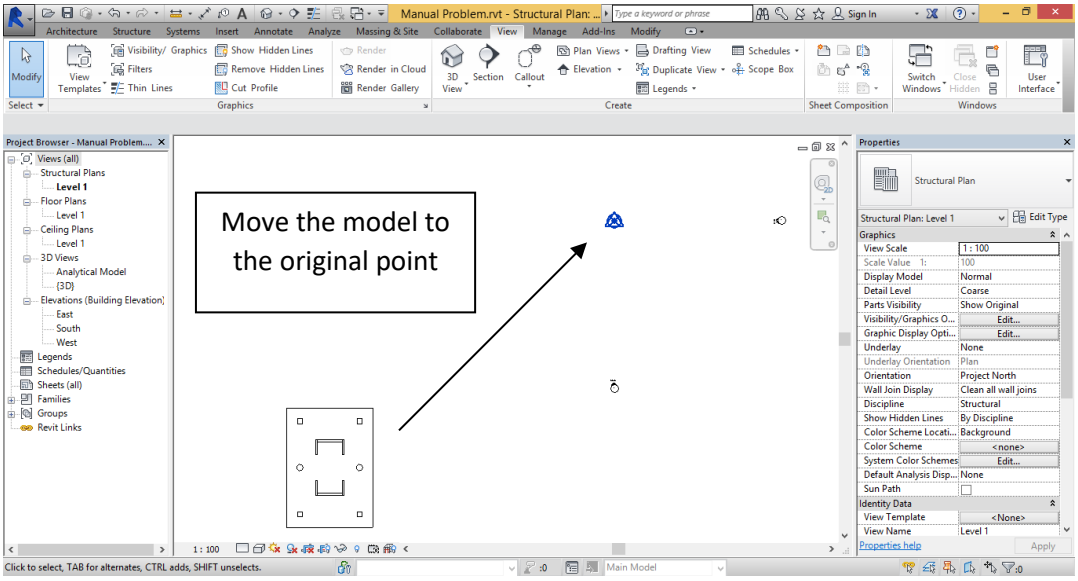


Figure (25): location of the import .dxf file in Autodesk Revit

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 **fully or partially located above the column** as shown in figure (26)

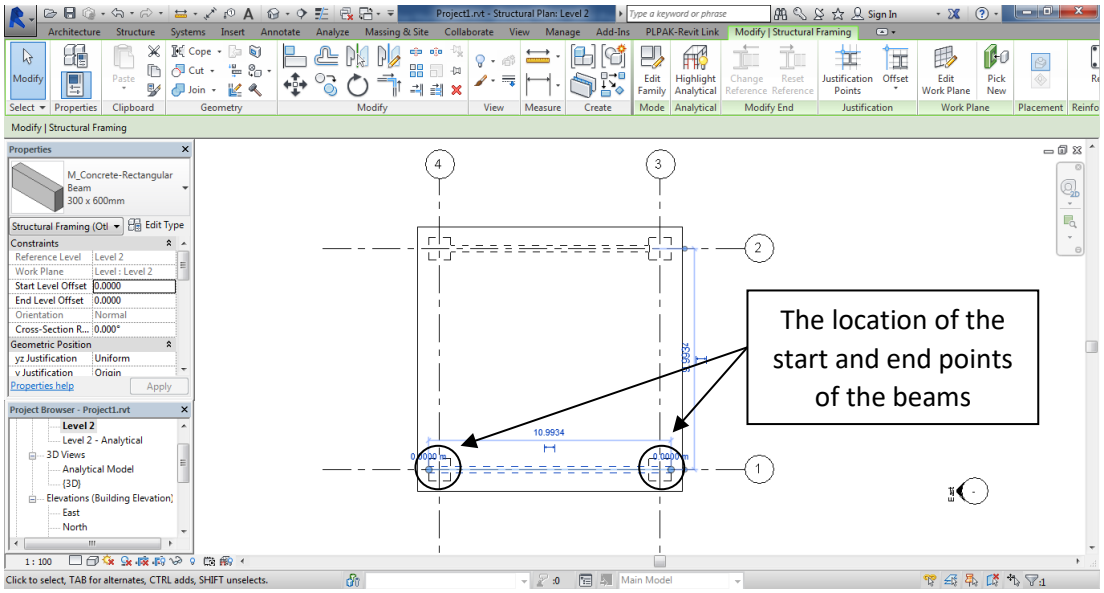


Figure (26): Simulation of fixed beam in Autodesk Revit

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).

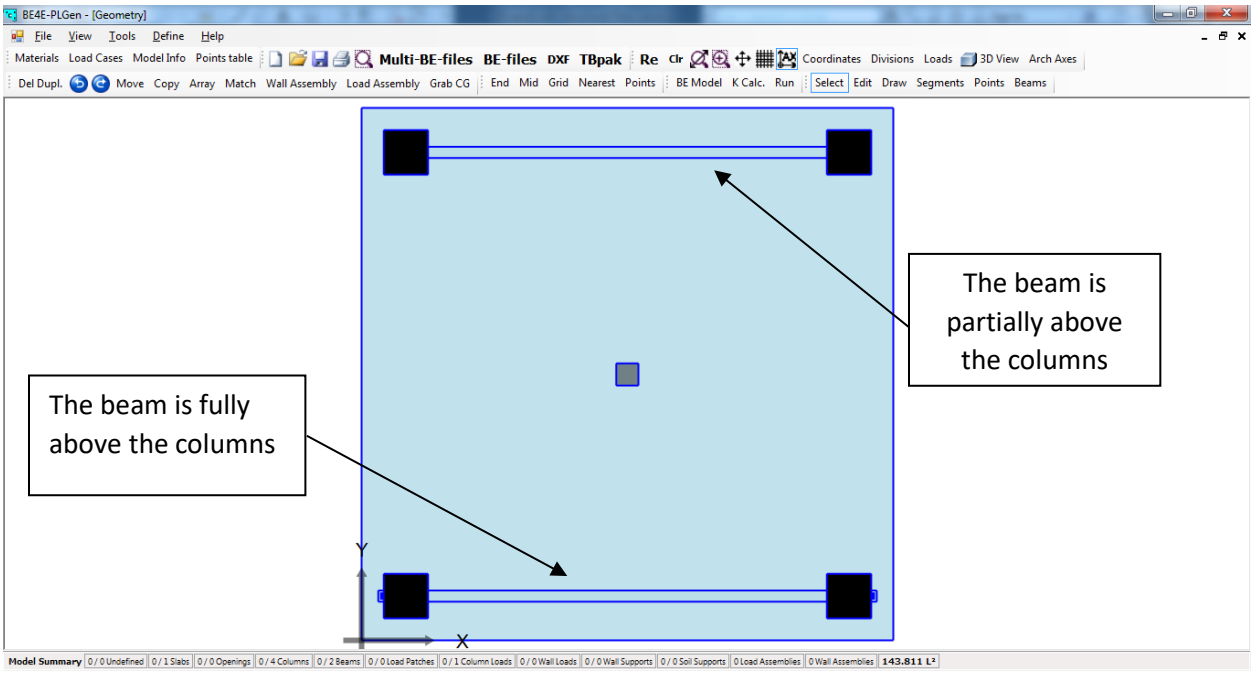


Figure (27): Simulation of fixed beam in PLGen.exe

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).

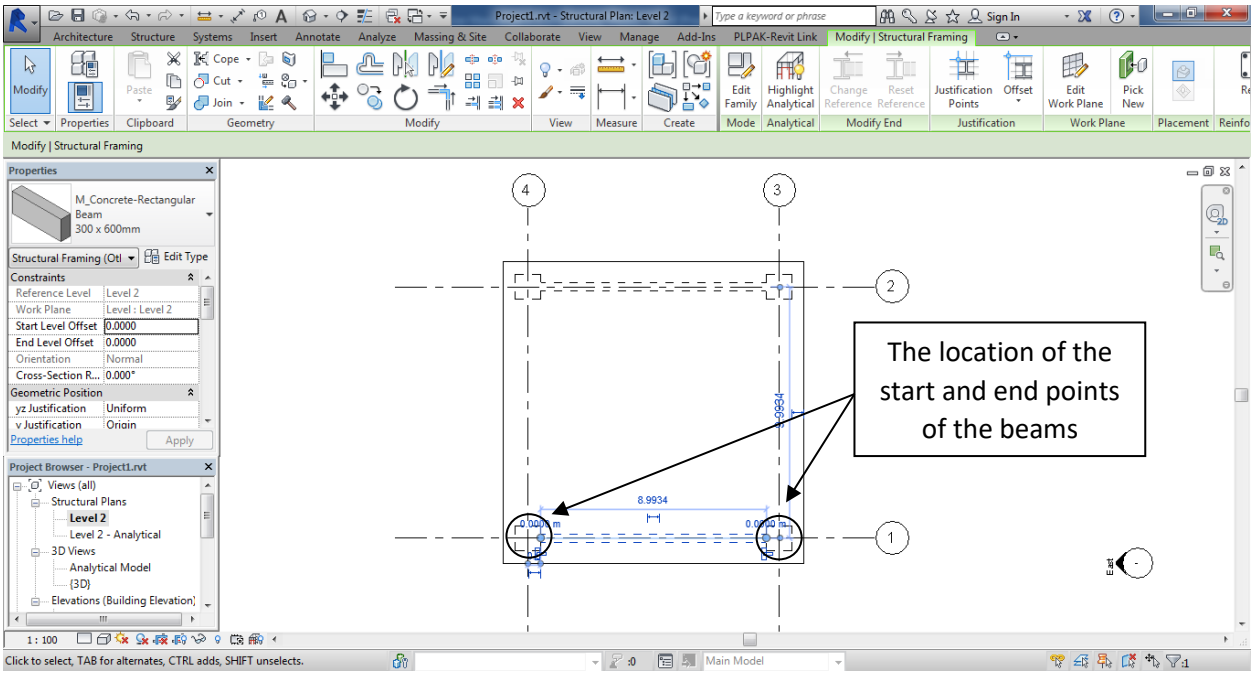


Figure (28): Simulation of released beam in Autodesk Revit

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).

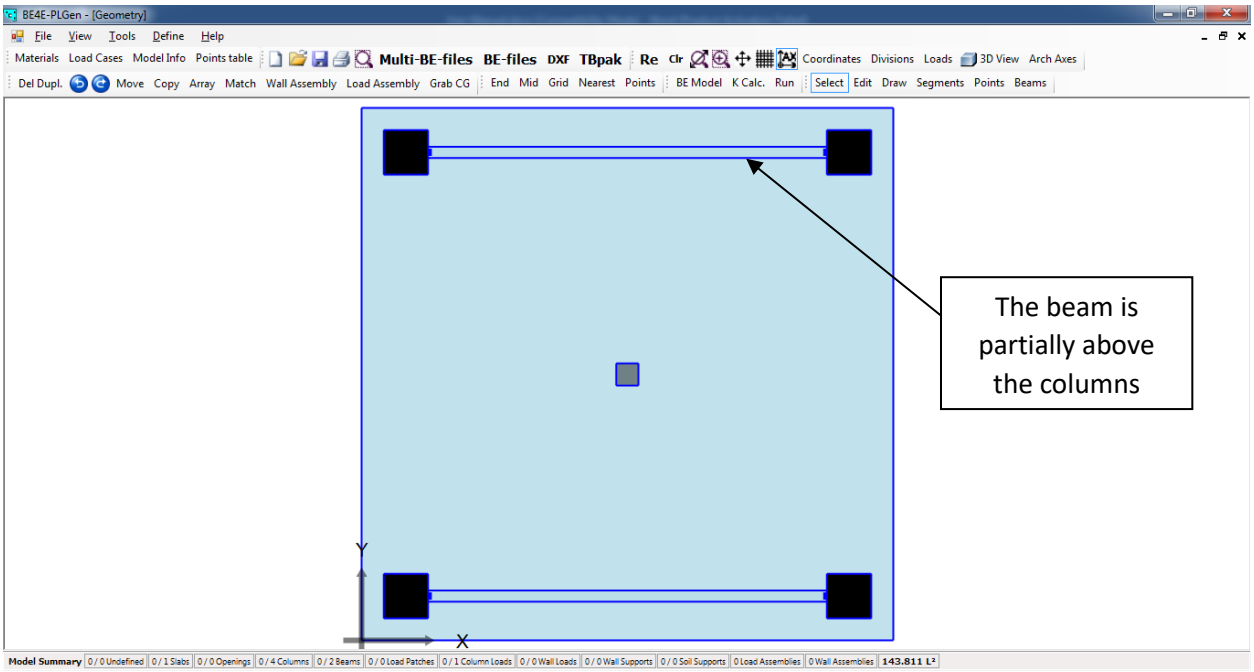


Figure (29): Simulation of released beam in PLGen.exe

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).

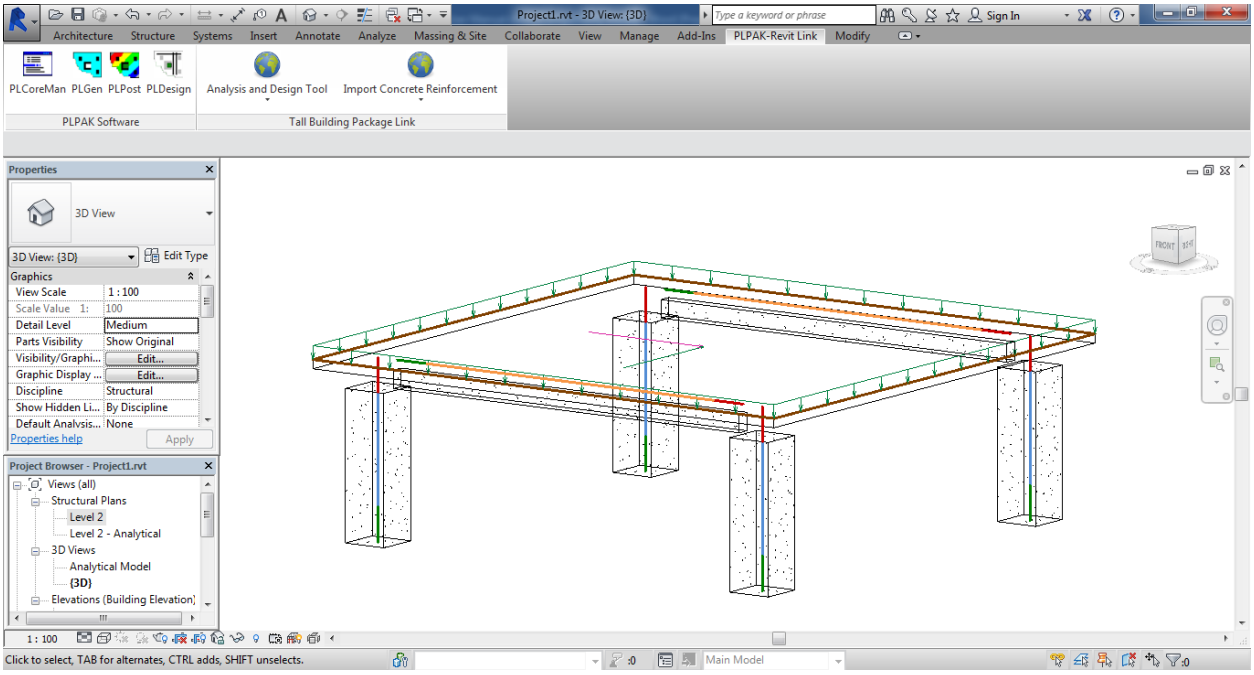


Figure (30): 3D view for beam column model

To change column height at the same time, keep the analytical line as it is:
In 3d view select all model ⟶ filter ⟶ check none ⟶ check on structural columns ⟶ Detach top/bottom ⟶ Detach all ⟶ select each column ⟶ change top offset ⟶ select all model ⟶ filter ⟶ check none ⟶ check on analytical columns ⟶ 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)

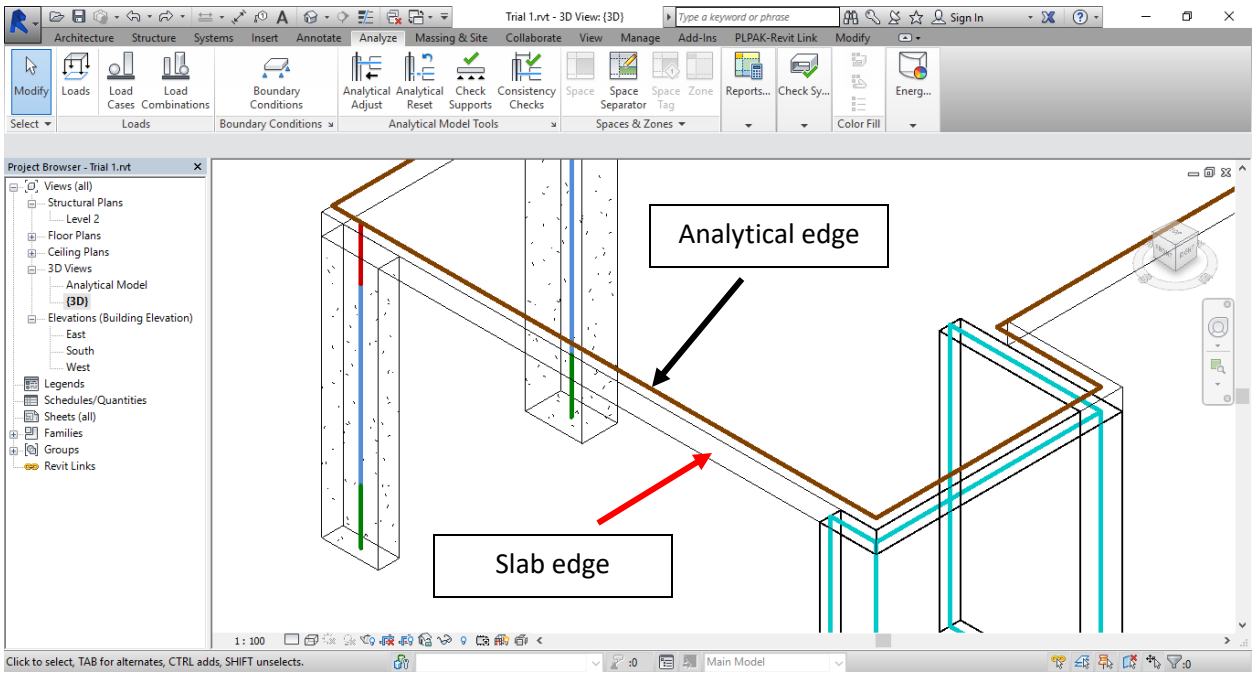


Figure (31): The location of slab edge and the analytical model.

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.

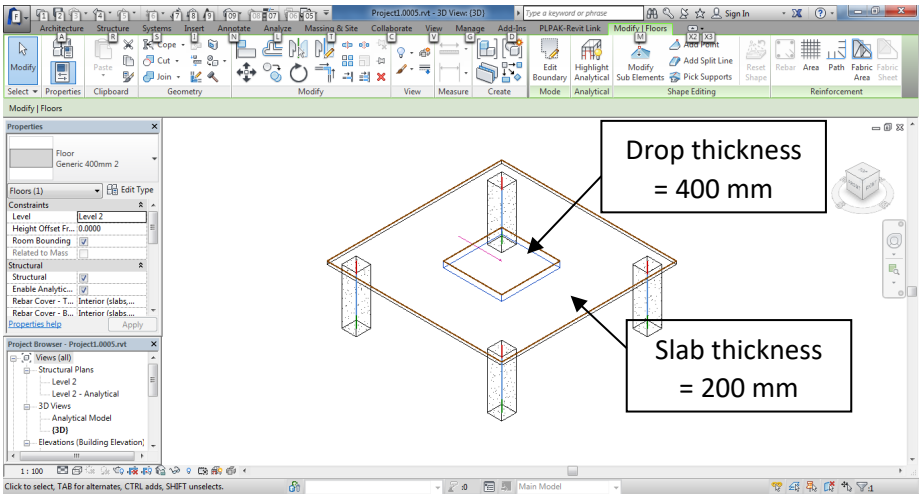


Figure (32): 3D view for slab drop model

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).

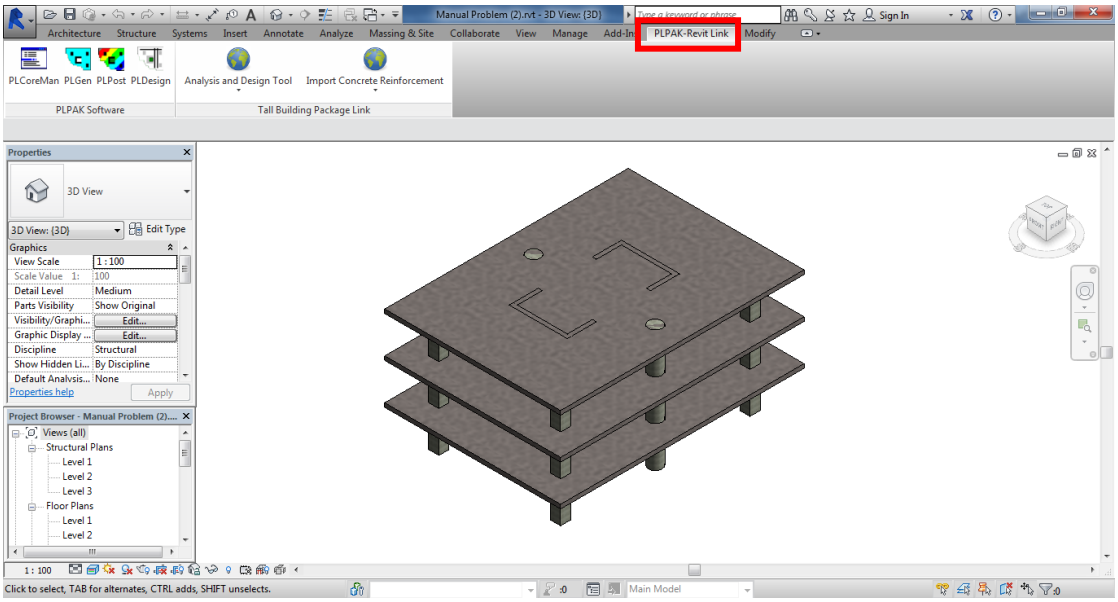


Figure (33): PLPAK-Revit Link

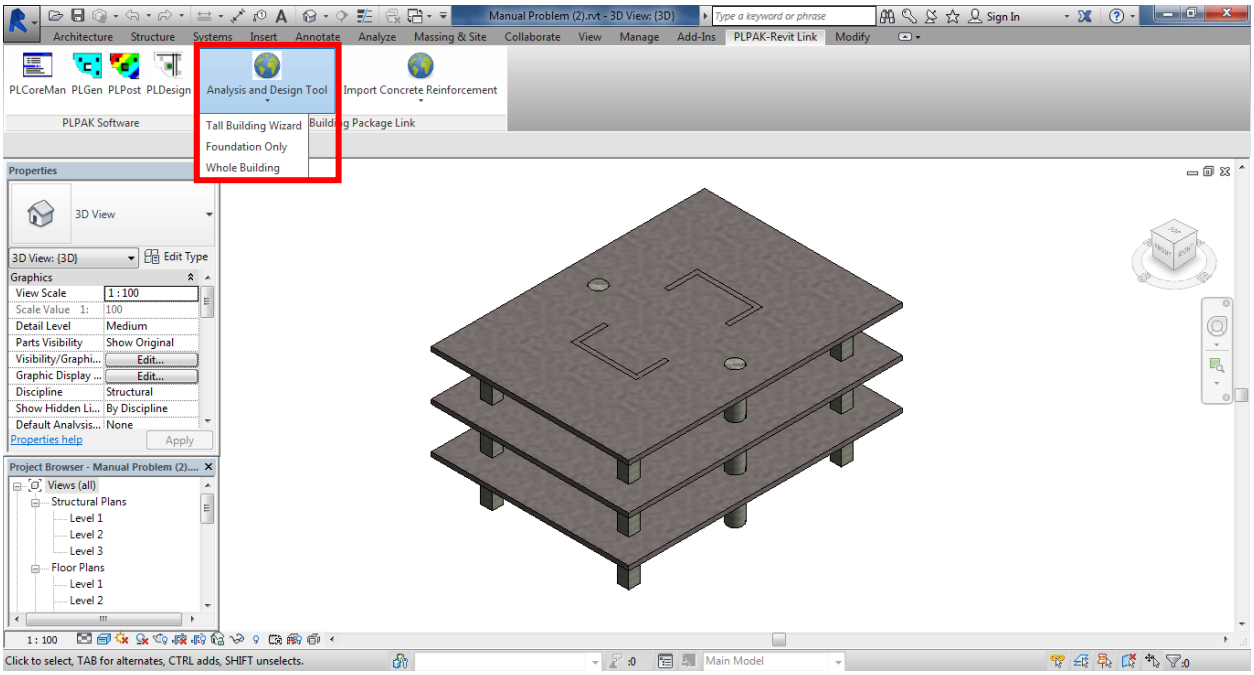


Figure (34): Analysis and Design Tool

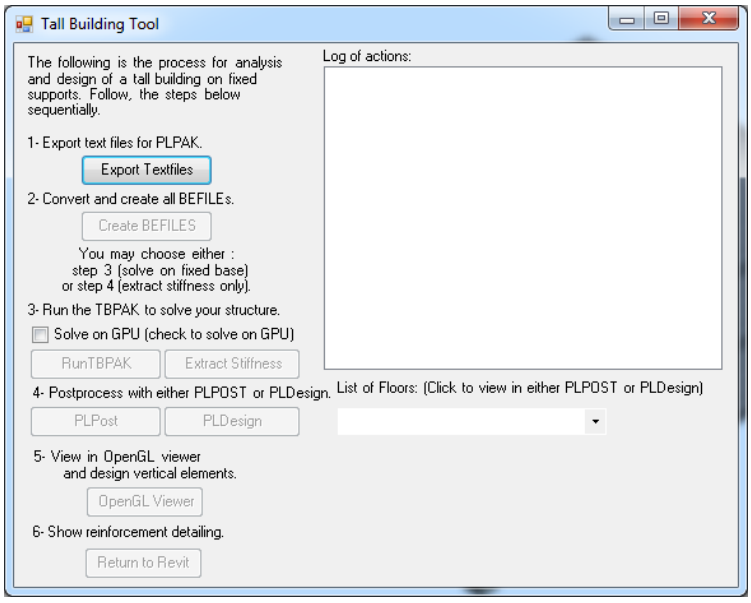


Figure (35): Tall Building Wizard

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).

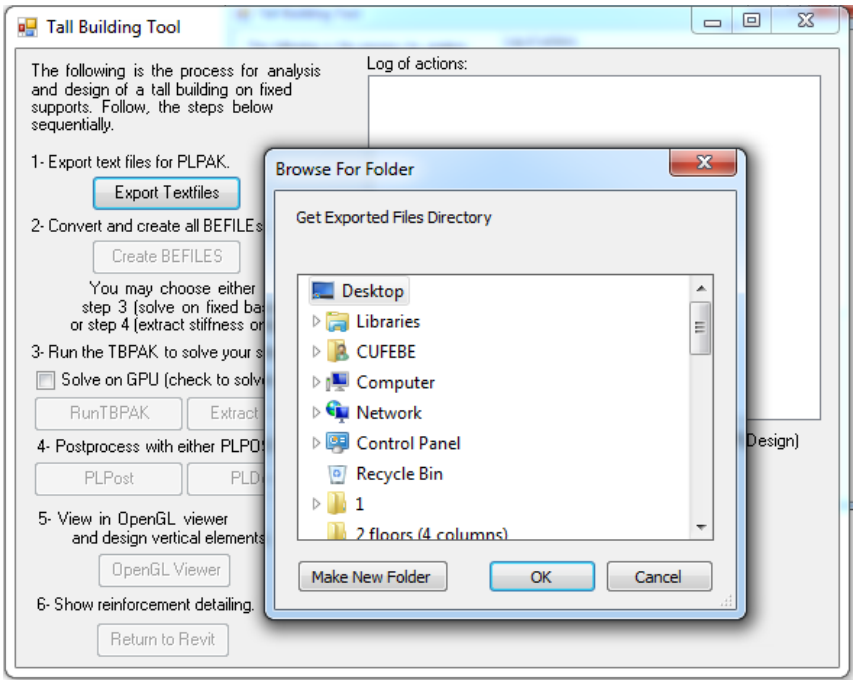


Figure (36): Export text files

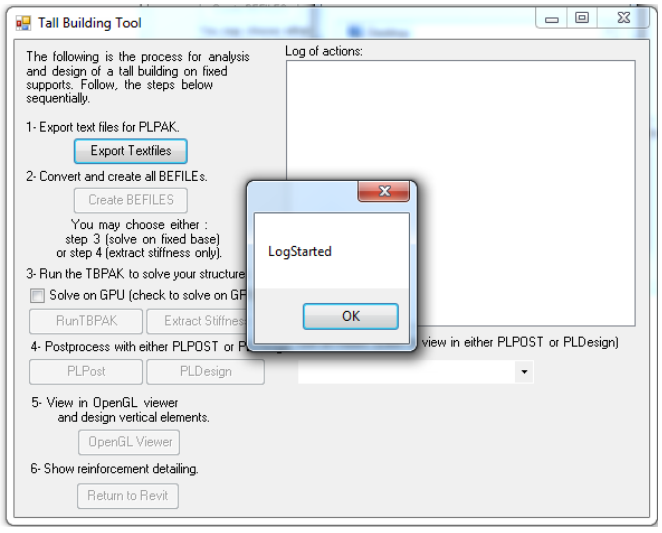


Figure (37): Start export text files

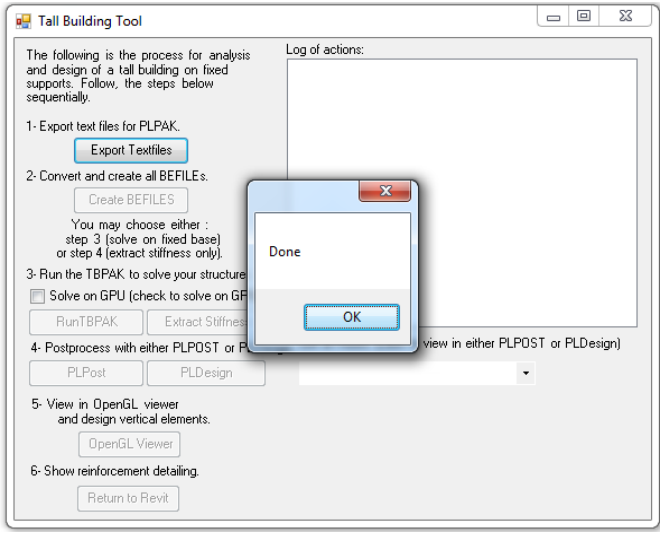


Figure (38): finish export text files

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

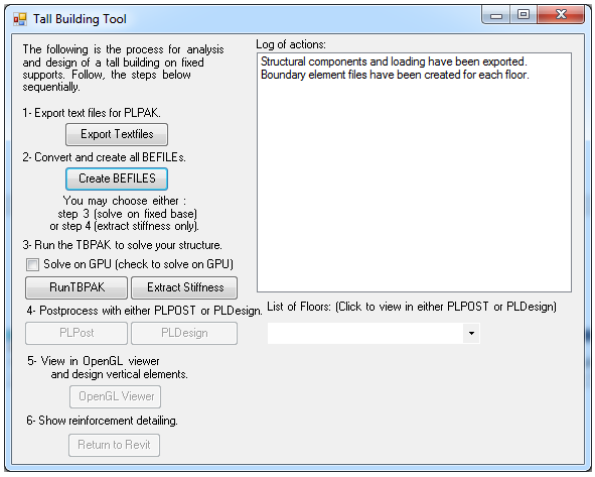


Figure (39): Create BEFILES

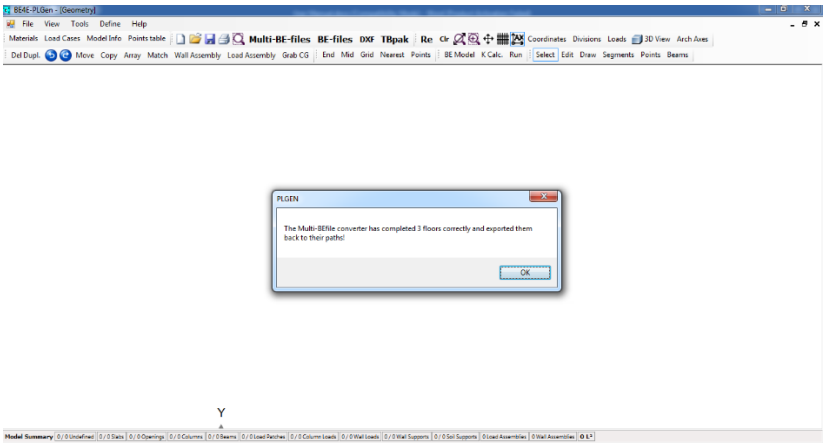


Figure (40): Finishing create BEFILES

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 ⟶ open each floor ID ⟶ open .gen file for each floor to check your model.

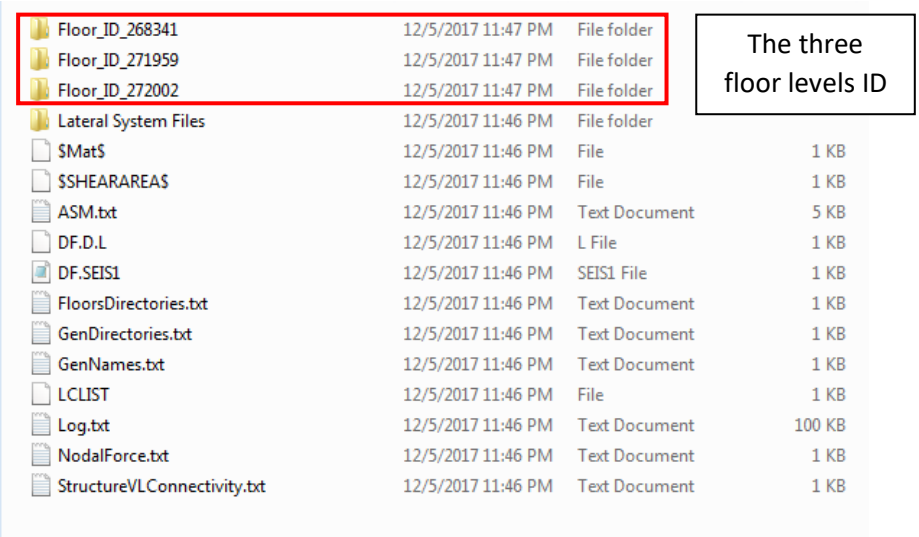


Figure (41): Floor levels ID

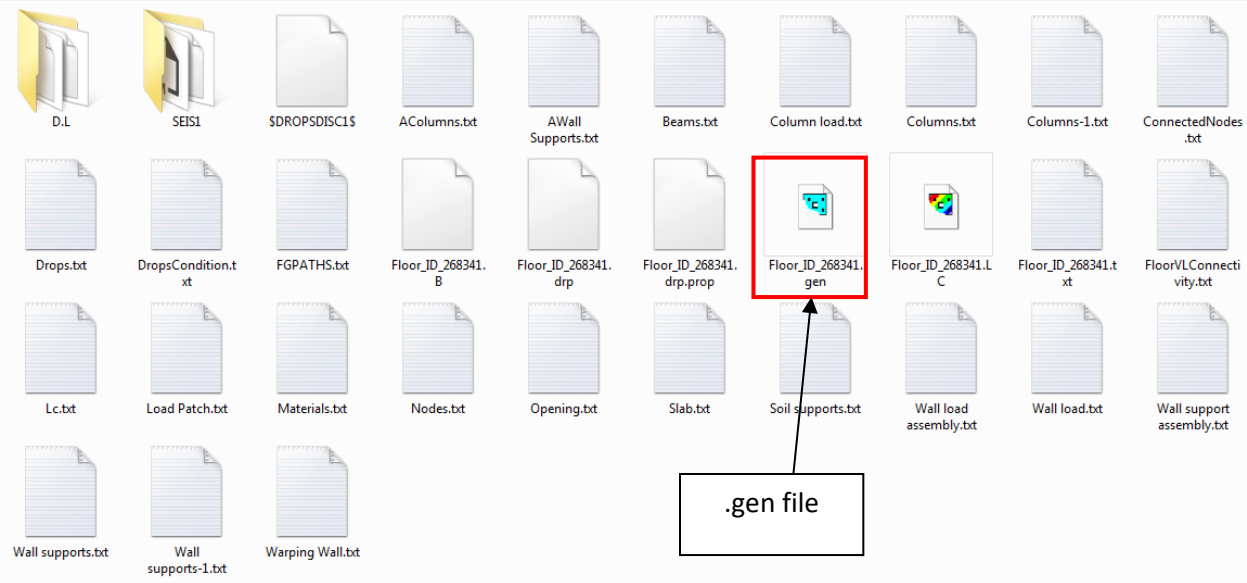


Figure (42): .gen file (PLGen.exe) for one floor ID

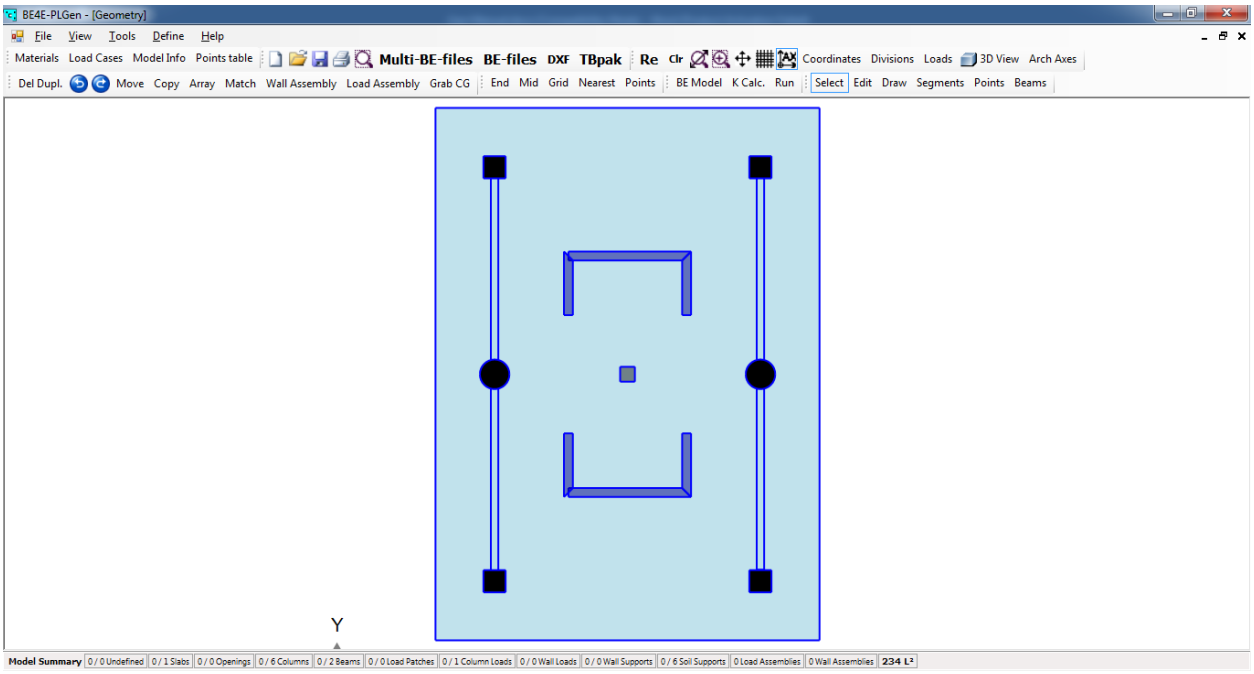


Figure (43): Boundary element model for each floor level

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).

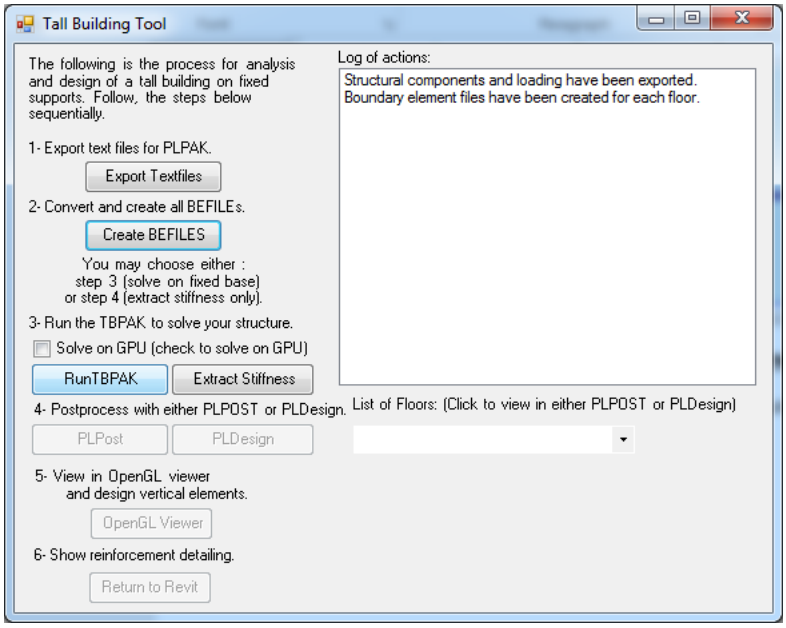


Figure (44): Run TBPAK.exe

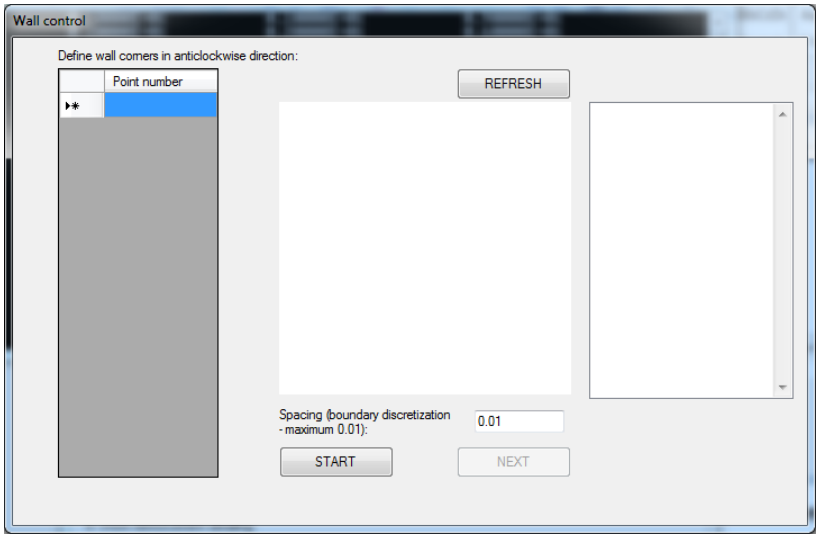


Figure (45): Wall control window

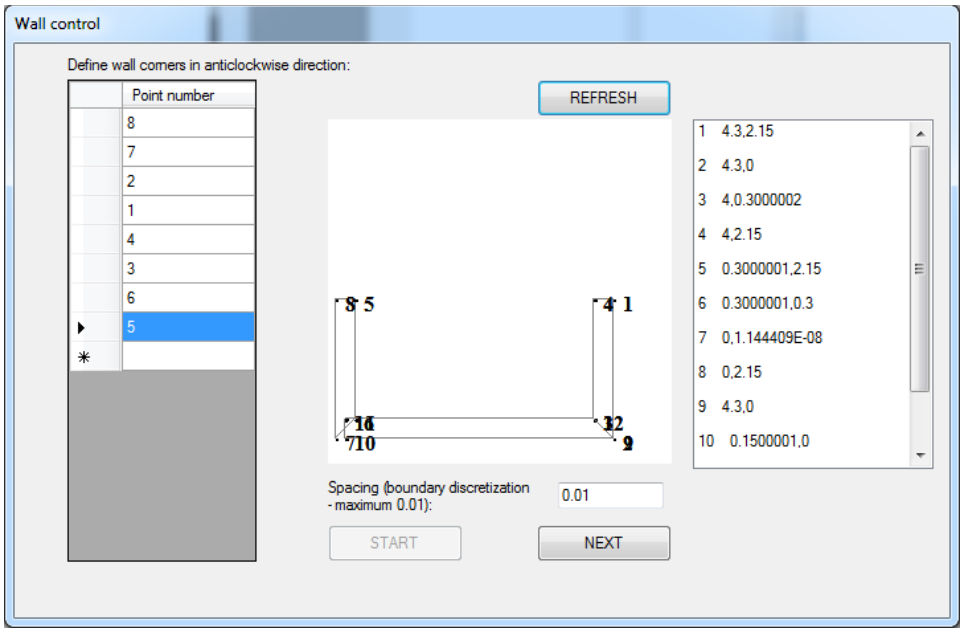


Figure (46): Write wall corners anticlockwise

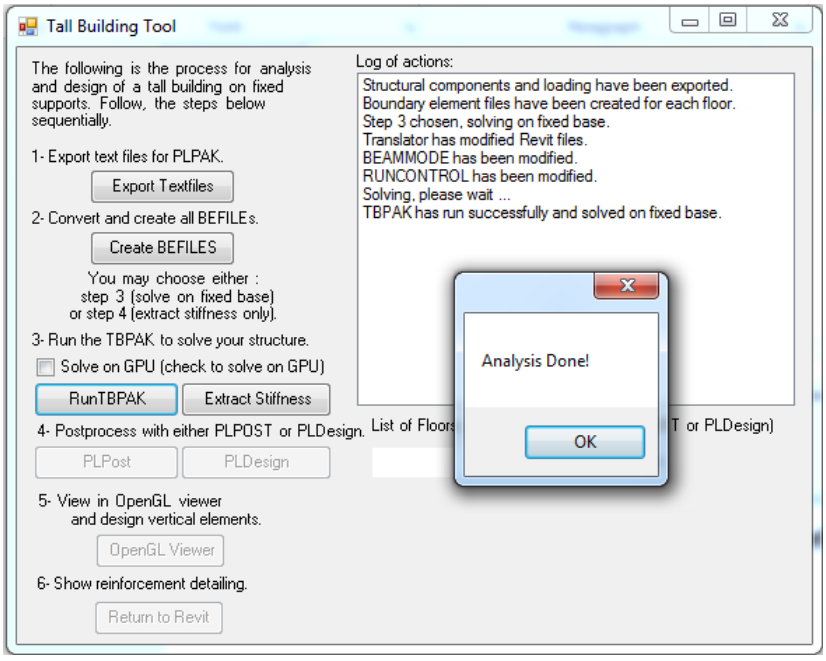


Figure (47): Finishing TBPAK.exe

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).

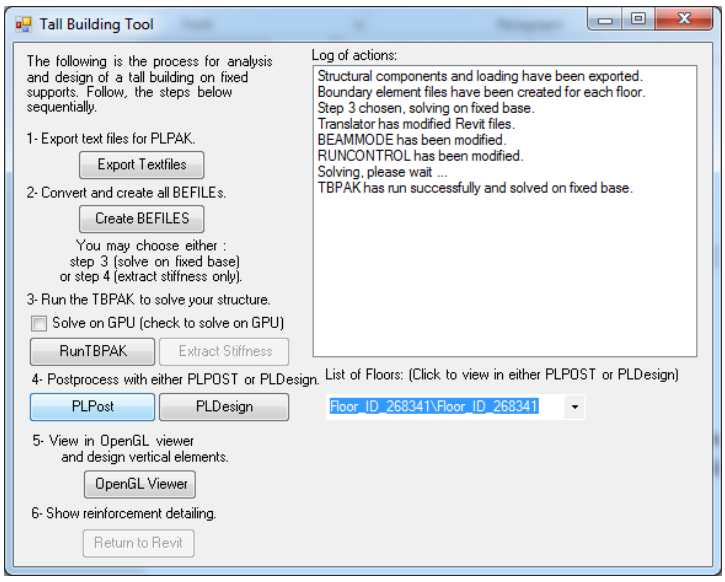


Figure (48): Open PLPost.exe

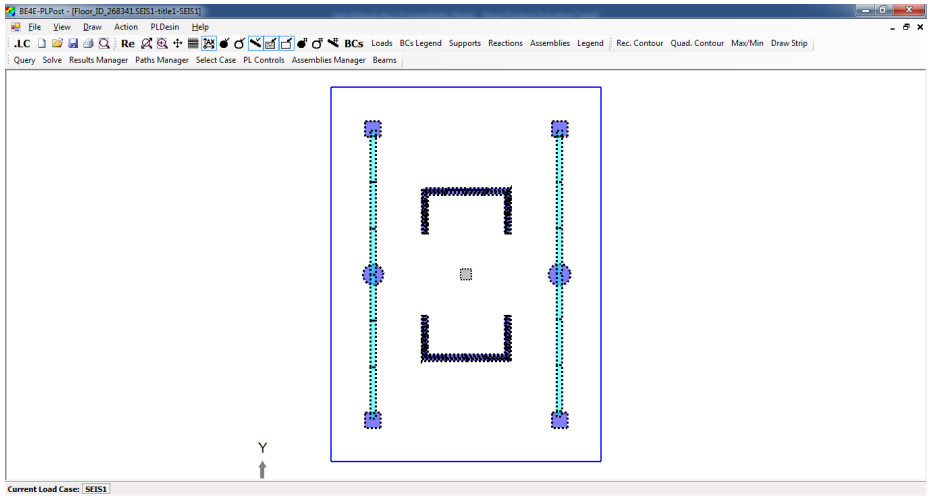


Figure (49): Open PLPost.exe

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

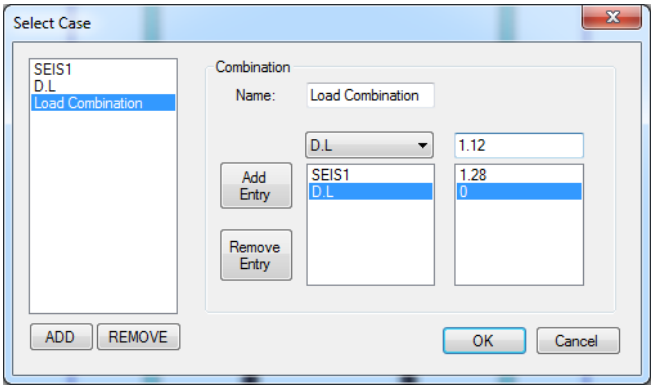


Figure (50): Load combination window

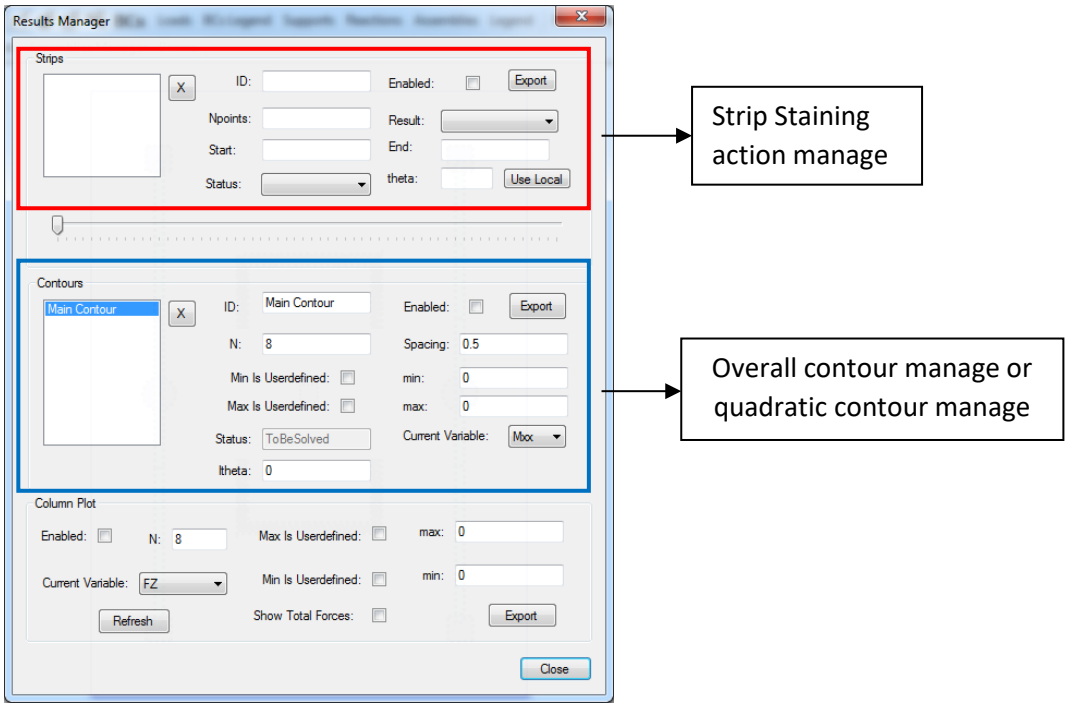


Figure (51): Result manger window

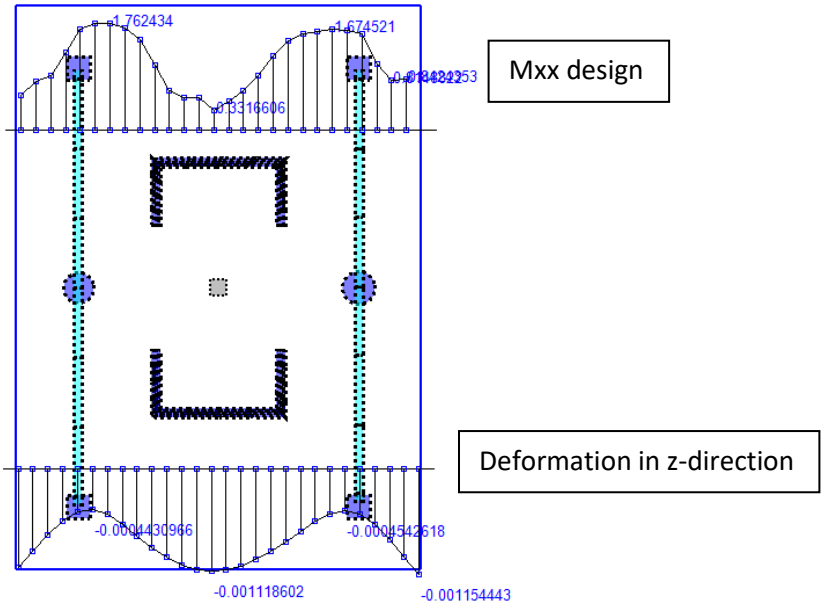


Figure (52): Strip Straining action

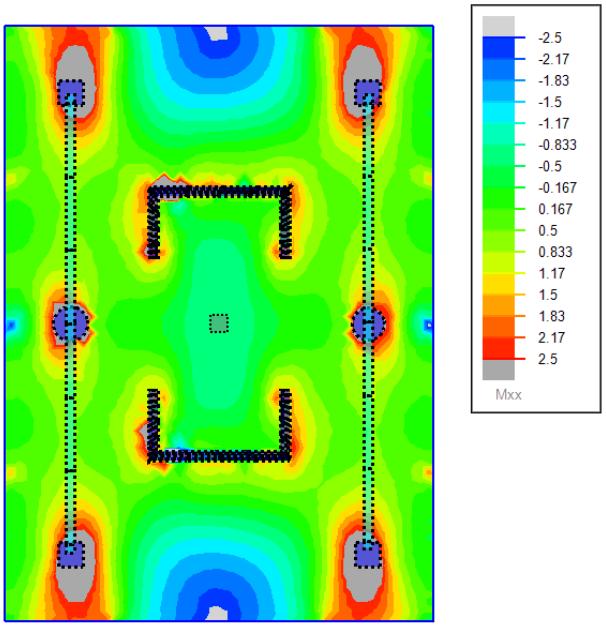


Figure (53): overall contour

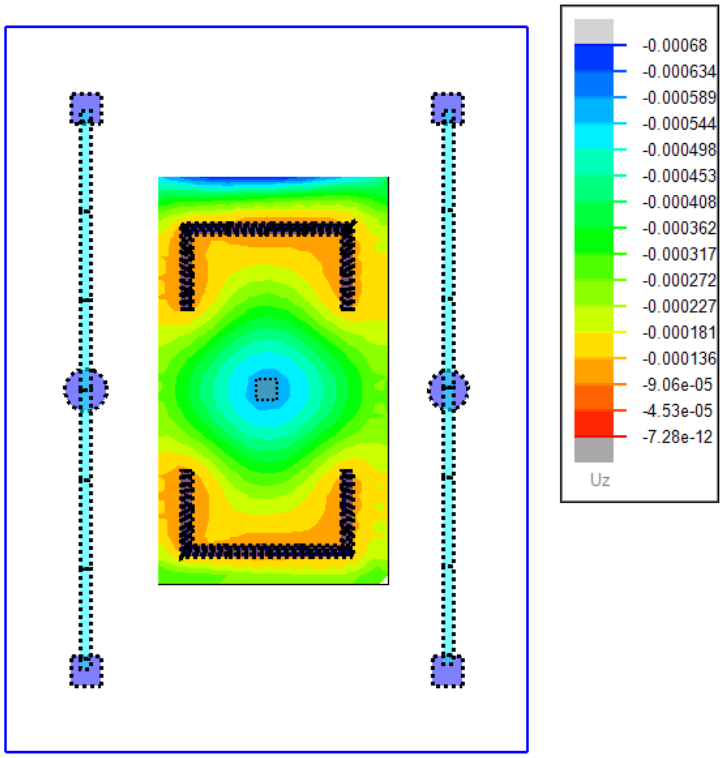


Figure (54): rectangular contour

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

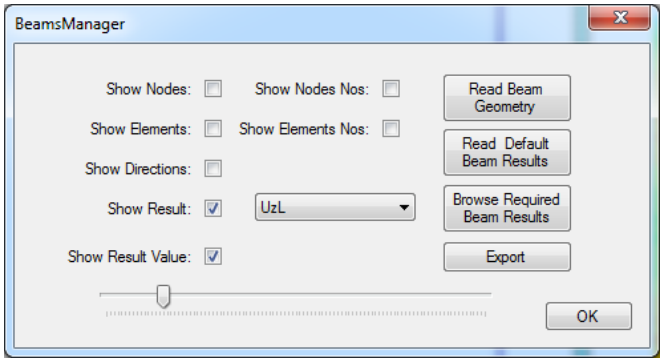


Figure (55): beams manager window

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).

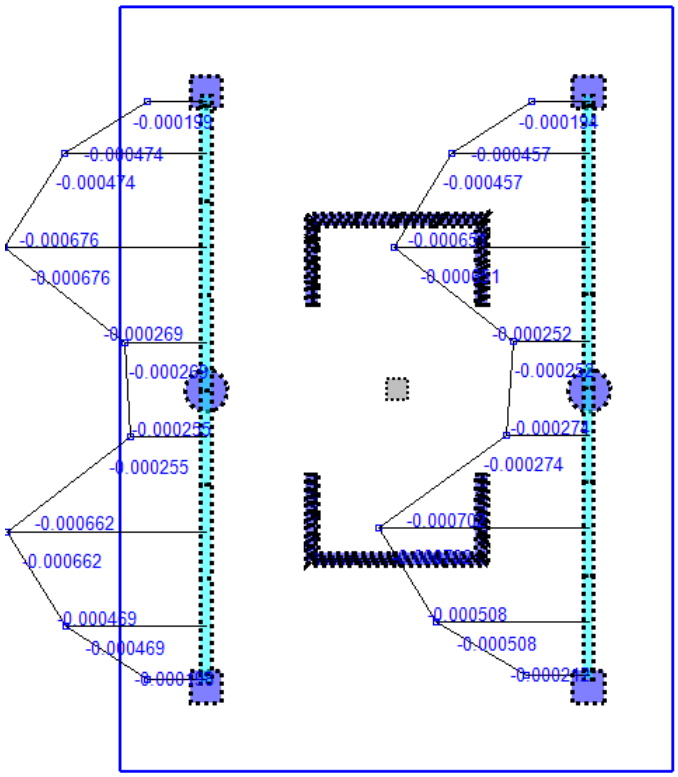


Figure (56): beams straining actions

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).

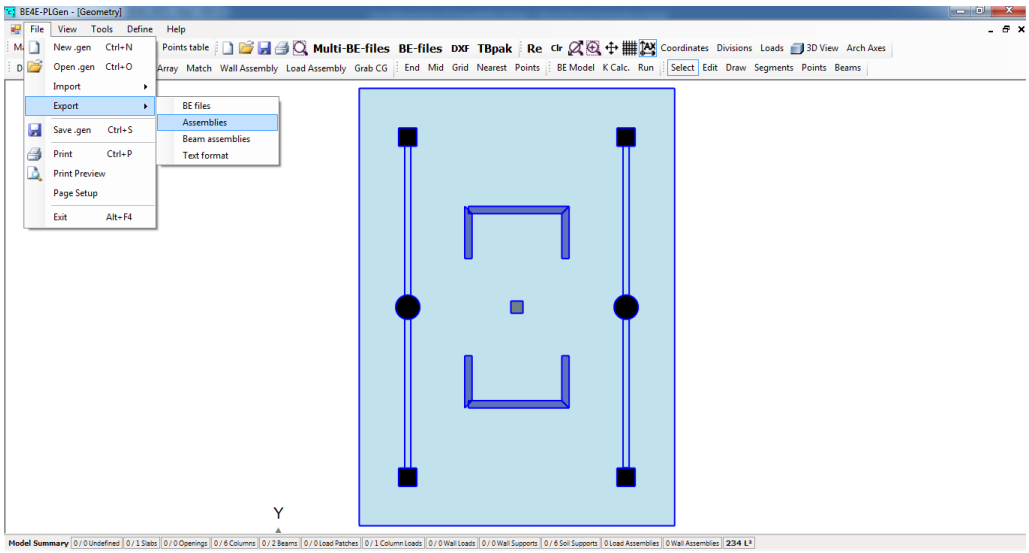


Figure (57): export assembly file

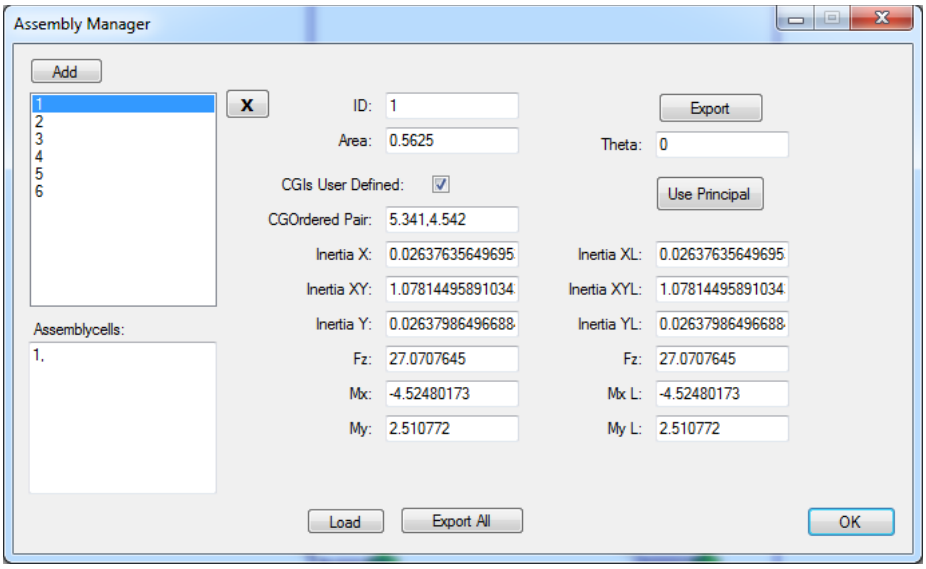


Figure (58): load assembly file

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).

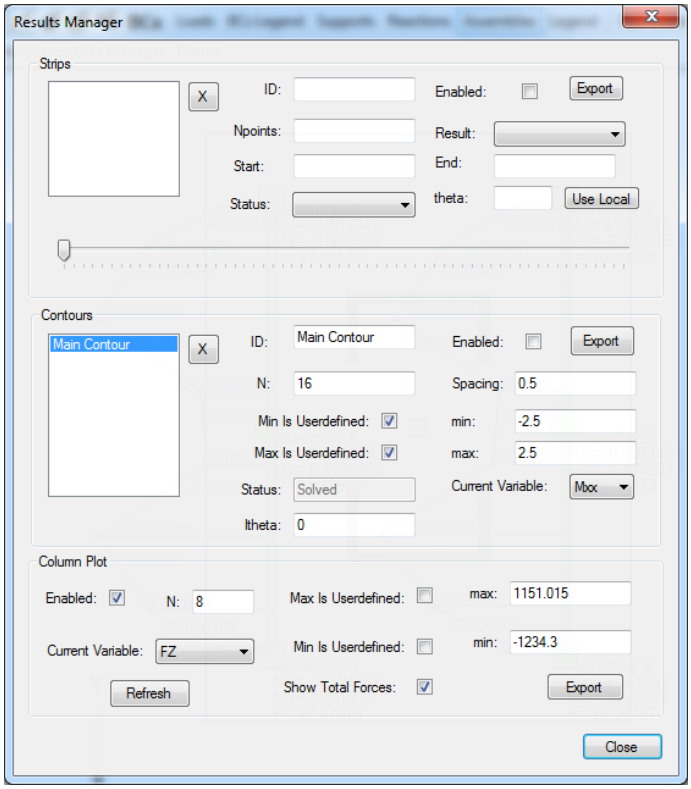


Figure (59): columns result manager

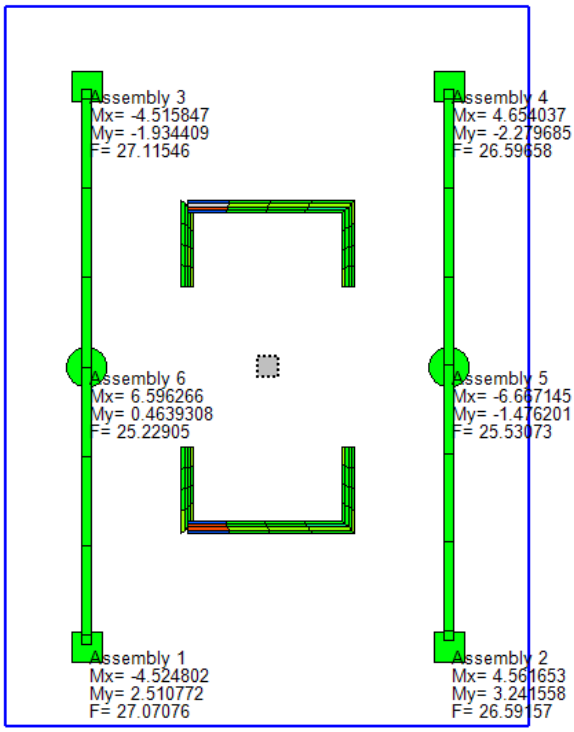


Figure (60): columns straining action

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 www.plpak.com regularly for *Problems and Solutions* section and the *Frequently Asked Questions* section