

The PLPAKTM

3D Viewer (OpenGL) Manual

PLPAKTM Version 2.00

**STRUCTURAL ANALYSIS SOFTWARE USING
THE BOUNDARY ELEMENTS METHOD**

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

E-mail: plpak@be4e.com

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.

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>

OpenGL Manual

- **Fixed Base Visualization and Vertical Elements Design**

The OpenGL tool is a tool used for the visualization and design of vertical elements; Figure 1 showcases an overview of the tool. This tool is initialized by importing the floor group file (\$FG\$), this floor group file is located at the PLPak folder in the 'program files' directory.

The (\$FG\$) file contains the data required for loading geometry and other structural information of the building into the OpenGL tool. The selection of the (\$FG\$) file is presented in Figure 2.

After loading the building, operational buttons appear and there are two viewing options for the building. the first option is the extruded view of the building, and the second option is the wireframe view of the building. Both views are presented in Figure 3 (A, B) respectively. The extruded view of the building indicates the sizes and location and orientation of vertical elements including columns shear walls and cores.

The Wireframe viewing style is used to visualize the straining actions applied on the vertical elements, by selecting elements and points, the user can view the loads applied. Also, view the deformed shape and view at the design options for the vertical elements.

The change between the extruded view and the wireframe view is conducted using the 'Toggle Drawing Mode' presented in Figure 4. The wireframe view enables several options for visualization. The user can start visualizing the straining actions using the drop-down menu where the load cases can be shown Figure 5. This drop-down menu includes all the load cases. Selecting a loadcase loads the corresponding string actions and deformations resulted from the PLPak analysis.

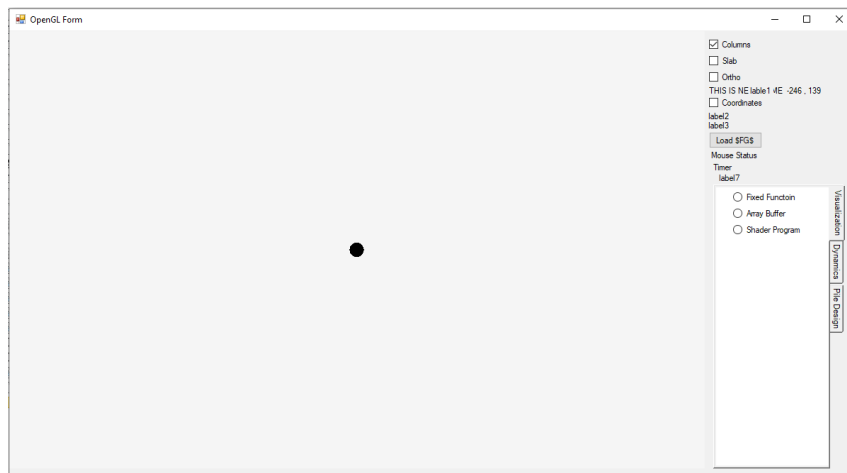


Figure 1. Overview of the OpenGL tool.

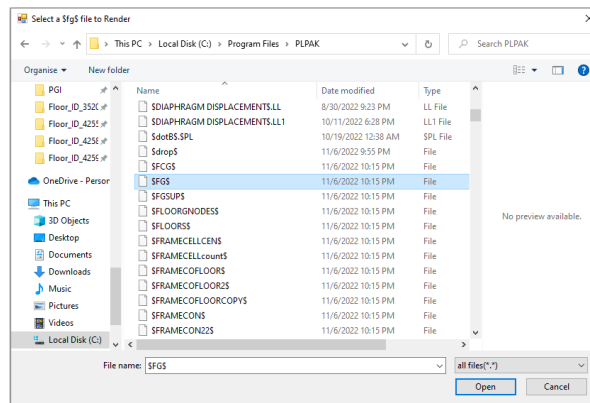


Figure 2. \$FG\$ file selection.

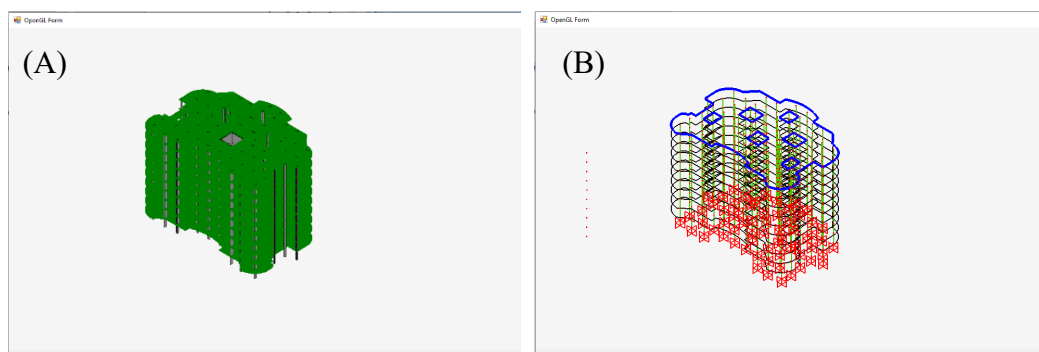


Figure 3. (A) the extruded view of the building, (B) The wireframe view of the building.

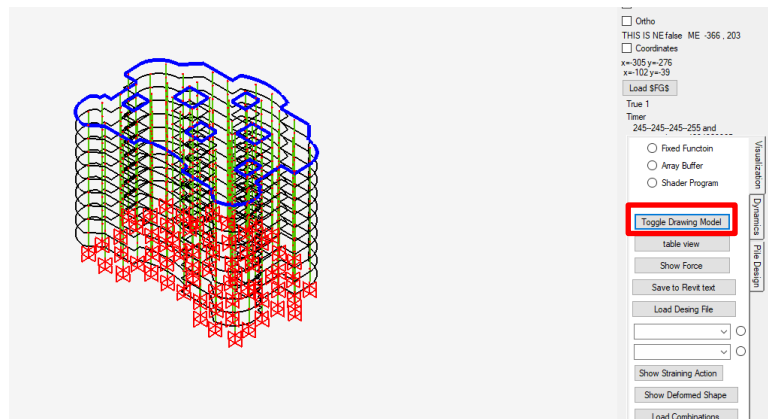


Figure 4. The ‘Toggle Drawing Model’ Button.

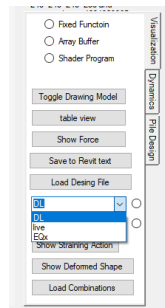


Figure 5. Drop down menu for the loaded load cases.

- **Loadcase information (applied load, deformation, and straining actions) visualization.**

To visualize the deformed shape as presented in Figure 6. it can be displayed for the selected loadcase using the show deformed shape button. the user chooses the scale of the deformed shape. Selecting a point in the deformed shape would display the value of the deformation, the tooltip for deformation values is presented in Figure 7.

The user can show the applied forces on any element by clicking show apply forces as illustrated in Figure 8. and the user can review all the point related the data for deformation and applied forces using the tabulated bottom the 'table view' button as presented in Figure 9.

To visualize the straining actions, the user can simply choose the type of string action and the scale of it and show string action applied to the vertical elements as presented in Figure 10.

After displaying the string actions on the vertical elements, a mouse right click on a vertical element will extend the viewers information to show the correspondence between straining actions. Showing the corresponding values can be done by simply hovering the mouse over the diagram as presented in Figure 11.

Load combination can be created using the load combination button and show the load combination form presented in Figure 12, the load combination form is used to load previously saved load combinations or generate new ones. The user generates new combinations by selecting the load cases and their corresponding load factors as illustrated in Figure 13. Two options are available for the combination, linear addition, or envelope.

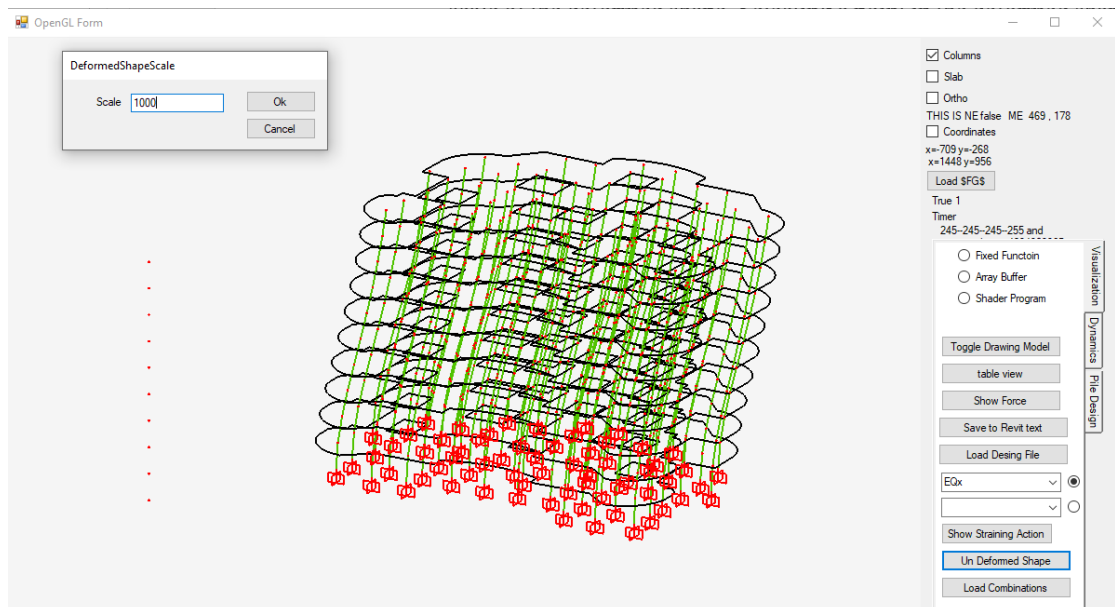


Figure 6: Deformed shape and Deformed Shape Scale form.

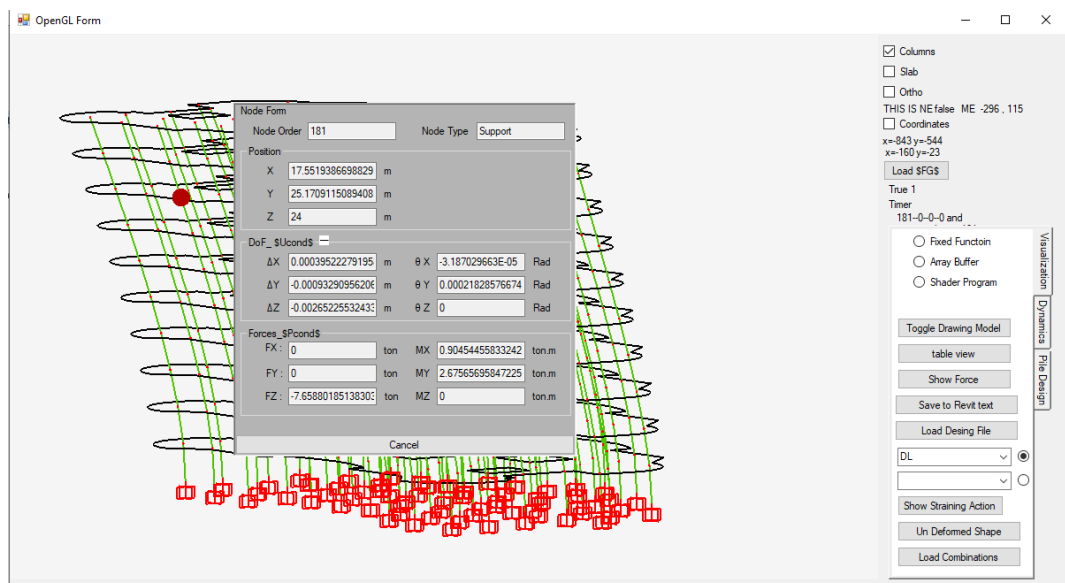


Figure 7: Deformation Values after Right Click on a Deformed Node 'DoF_\$_Ucond\$'.

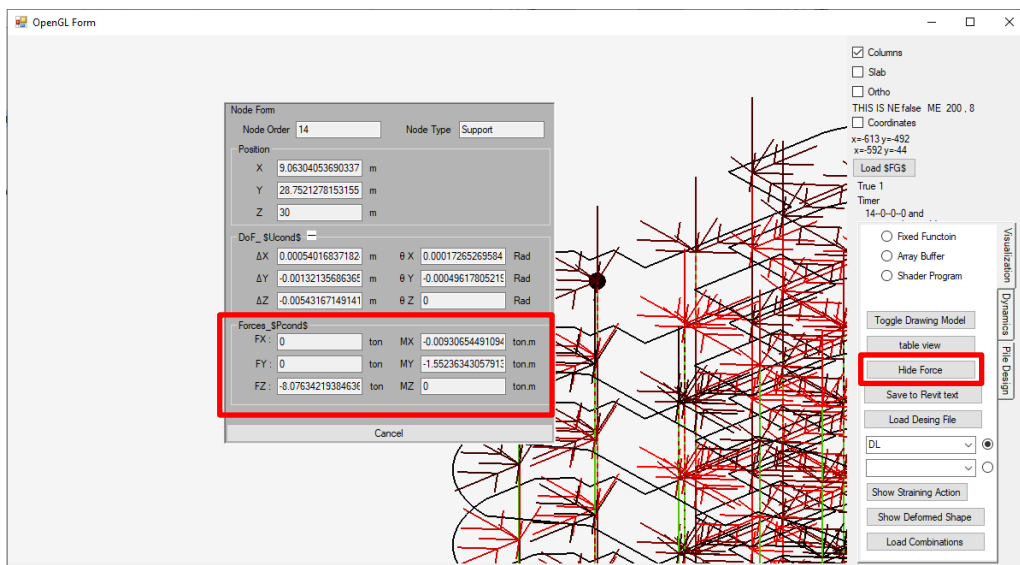


Figure 8: The 'Show/Hide forces Button' and Right click on Node to show Applied Force values.

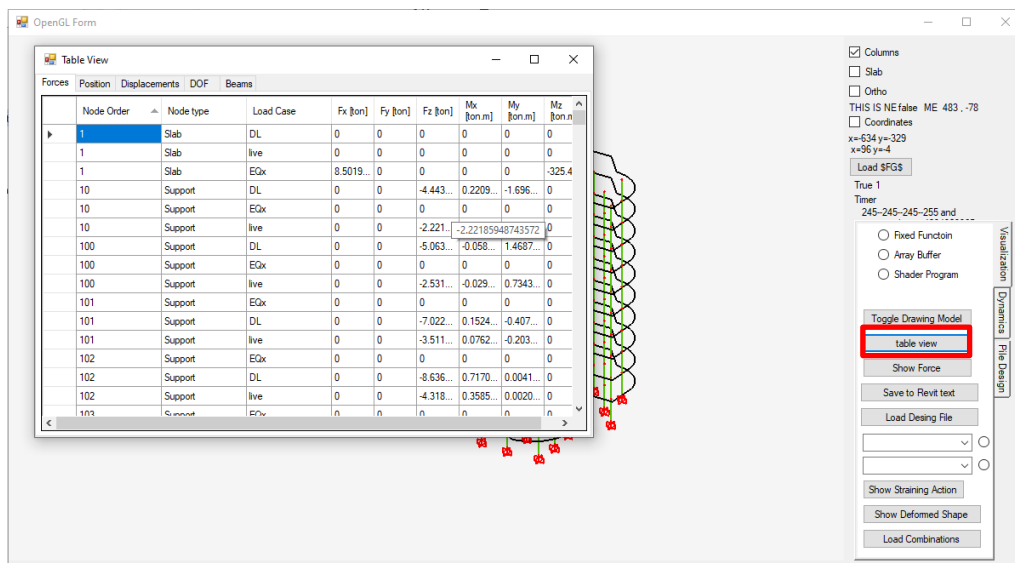


Figure 9: Tabulated Data View and 'table view' button.

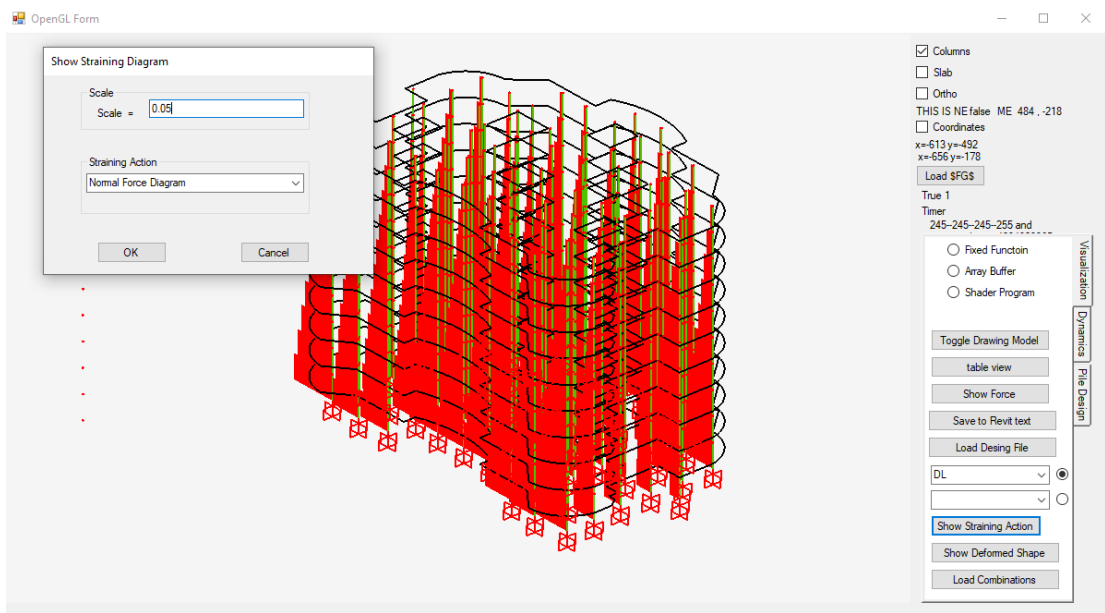


Figure 10: The Straining Action Diagram for Normal Stresses Scaled to 0.05.

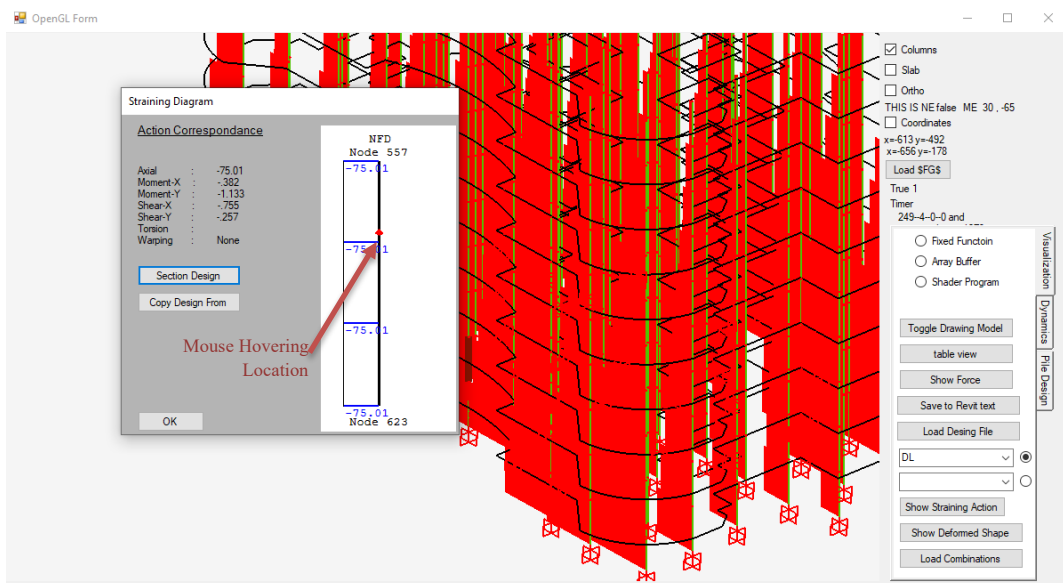


Figure 11: Straining Action Form with Corresponding values option.

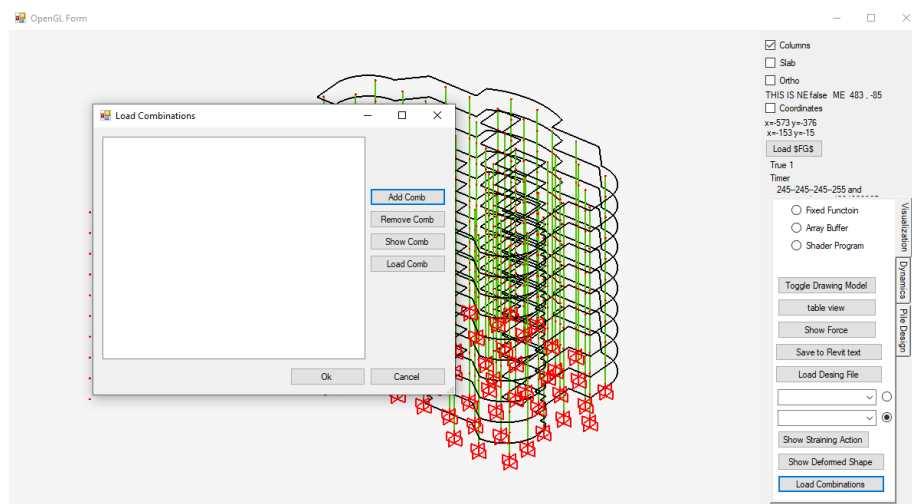


Figure 12: Straining Action Form with Corresponding values option.

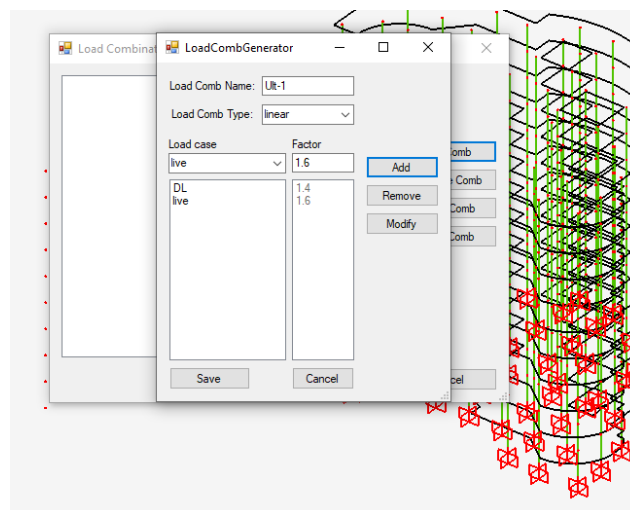


Figure 13: Straining Action Form with Corresponding values option.

- **Reinforced concrete section design.**

From the string action viewing tool, the user can start designing the vertical elements. choosing to design A vertical element opens a new window shown in Figure 14, this window includes the cross section of this vertical element and text boxes and entry points for the properties of material used in the design steel yield stress for a vertical bars and concrete compressive strength.

By filling materials properties data and choosing the number of bars and rebar diameter and concrete cover value, the user can add a line of rebar by simply clicking the edge off the concrete section as shown in Figure 15. The user can also use zooming tools to view a part or whole of the section (Figure 15 - box A) also the user can use the mouse scroll tool to zoom in and out and click and hold the mouse right button to pan the view of section. A single bar can be added to the section (Figure 15 - box B) also there are moving tools to adjust the placement of rebar reinforcement (Figure 15 - box C) Note that the motion for rebars is in millimeters and the text boxes for X and Y motion appear after the user clicks 'Move Bars', the user then left click on the desired bar.

The check of the stirrups count can be performed using the stirrups check bottom. The user inserts the spacing between stirrups and the number of stirrups legs in both direction X&Y (Figure 15 - box D). Then the user is presented with the safe stirrup diameter in each direction. The 'Add link bottom can be used to place stirrups for the section the user can simply select two bars as illustrated in Figure 16.

The interaction diagram for the section and the task of checking if moment capacity of the section exceeds the applied ultimate moment. This can be performed through the interaction diagram results by using CDesign software by following the steps illustrated in Figure 17 and Figure 18. the next step the user can view the interaction diagram generated using the 'show interaction Metric' button. A new Windows form will appear was four tabs, each tab represents a direction for moment capacity both positive and negative for both X&Y as presented in Figure 19. A cross mark can indicate whether the section is inside the interaction diagram or outside the interaction diagram for further reinforcement assessment.

After performing check of column reinforcement design the user can save the reinforcement to the section also the user can copy this reinforcement to another element by clicking 'copy enforcement from' shown in Figure 20, then left clicking the new element for reinforcement transfer.

To save all reinforcement generated in this model end to export reinforcement to rivet the user can click the save reinforcement text to Rivet bottom as illustrated in Figure 21. The 'Save to Revit text' button also saves the design in a '*.cdf' file format for elements to be loaded by the program as well.

Section Design

Material

Fc' in MPa Clear Rebar

Fy in MPa

Reinforcement

No of Bars / Row Select Edge Quick bar

Cover in mm Diameter in mm

Generate Text Input and Run CDesign Show Interaction Metric

Kill design Files Show CDesign Text Output

Move Bars Status : ---

Shear stirrup Design : Check Add Link Clear Stirrups

Spacing mm Chosen Dia. : mm

X direction Y direction

No. of links : No. of links :

Proposed Diameter : mm Proposed Diameter : mm

Save Cancel

(40.43, 25.17) (40.73, 25.17)

(40.43, 24.37) (40.73, 24.37)

Figure 14: Reinforced Concrete Section Design form.

Section Design

Material

Fc' in MPa Clear Rebar

Fy in MPa

Reinforcement

No of Bars / Row Select Edge Quick bar

Cover in mm Diameter in mm

Generate Text Input and Run CDesign Show Interaction Metric

Kill design Files Show CDesign Text Output

(C) Move Bars X mm : Y mm : Status : ---

(D) Shear stirrup Design : Check Add Link Clear Stirrups

Spacing mm Chosen Dia. : mm

X direction Y direction

No. of links : No. of links :

Proposed Diameter : mm Proposed Diameter : mm

Save Cancel

(A)

(40.43, 25.17) (40.73, 25.17)

(40.43, 24.37) (40.73, 24.37)

Figure 15: Section Design Form and options.

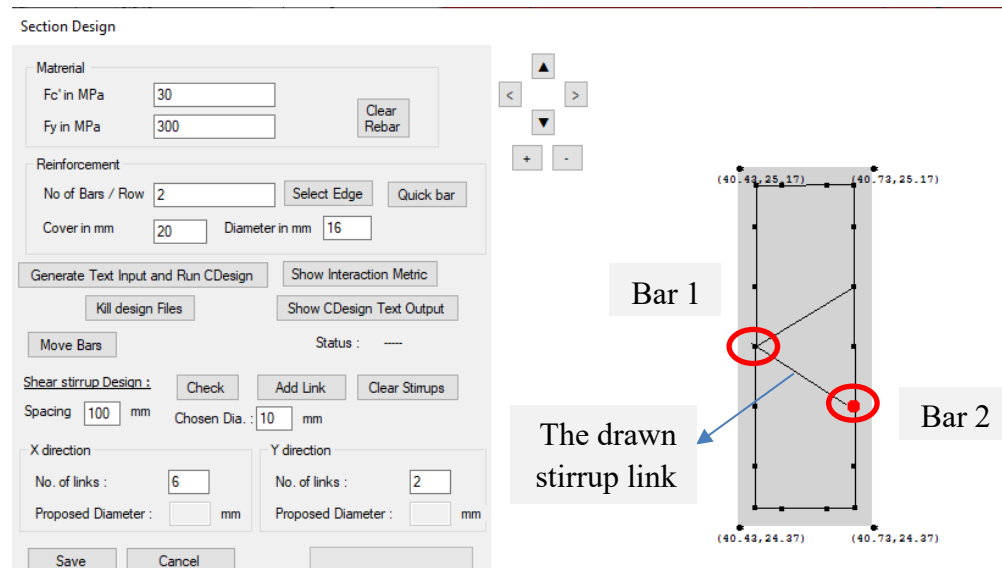


Figure 16: Stirrup link Drawing Between Two Bars ‘Note: red node on rebar while hovering’.

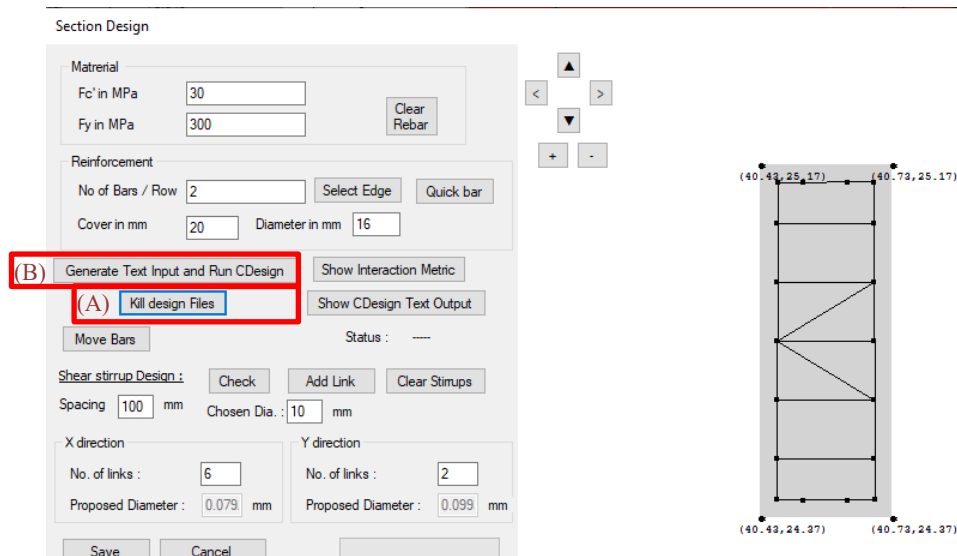


Figure 17: Step A: The User Removes Previous Results, Step B: The User Generates New Results.

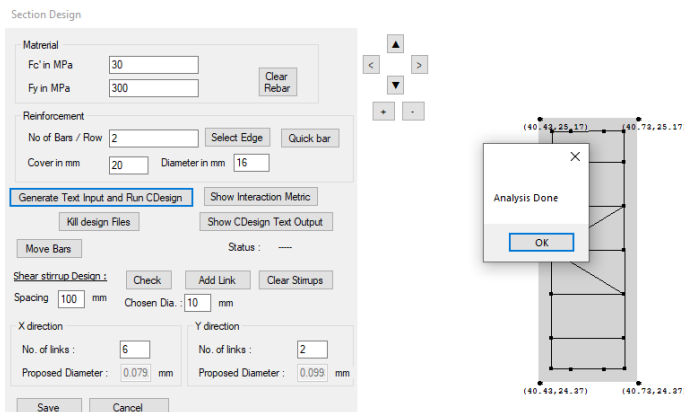


Figure 18: ‘Analysis Done’ Checkbox appears after Calculation of Interaction Diagram.

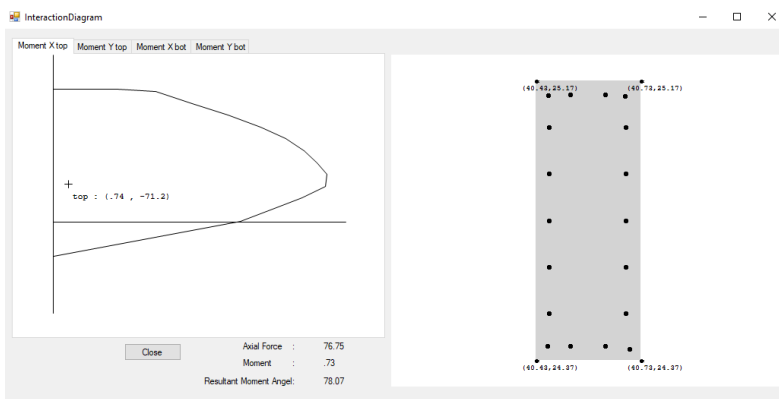


Figure 19: Interaction Diagram After Clicking ‘Show Interaction Metric’.

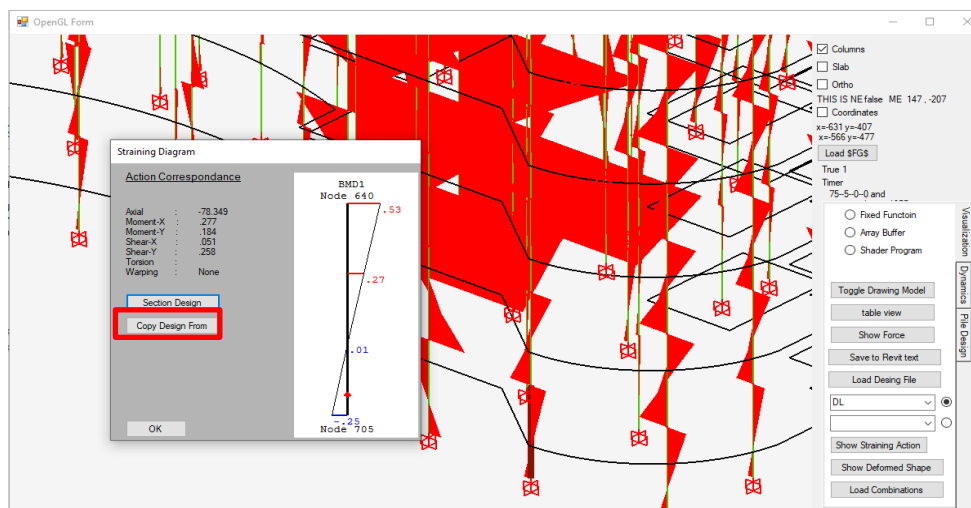


Figure 20: ‘Copy Design From’ Button is Used to Copy Reinforcement from One Section to Another (usually same section on different level).

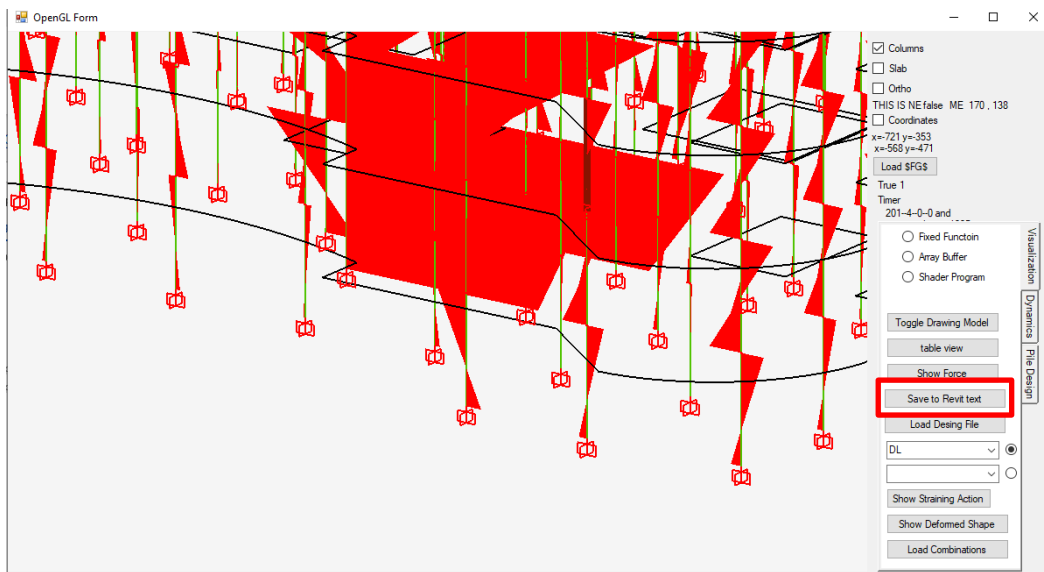


Figure 21: ‘Save to Revit text’ Button is Used to Generate Text Export to Revit.

- **Deep foundation design**

For the deep foundation design, and to view the foundation analysis straining action on piles, first, the user needs to change to the foundation design tab at the tap tool as presented in Figure 22.

Then by pressing the 'load raft file' button the user searches for the \$RaftFG\$ in the PLPak directory as showcased in Figure 23. Loading this file will reveal the floor plan for the foundation as presented in Figure 24, then the user loads the straining actions applied to the piles by choosing the load straining action and then choosing the 'columns.TSV' file presented at the problem directory as presented in Figure 25. After loading the 'columns.TSV' results file the piles would appear as presented in Figure 26.

For pile design, the user chooses a pile Loadcase from the loadcases drop down list presented in Figure 27 then by left clicking on a pile as shown in Figure 28 a design form would appear as presented in Figure 29.

Filling the data for the design to generate reinforcement for the pile and the form automatically calculates the factor of safety for the pile subjected to the string actions presented in the design form. By clicking 'save', the user saves the reinforcement including stirrups to their pile element, and then by clicking 'copy design to other piles' the user can simply copy reinforcement designed to other piles. The final step is to save the piles reinforcement and exporting them to Rivet by clicking the 'safe to river text' button as presented in Figure 30.

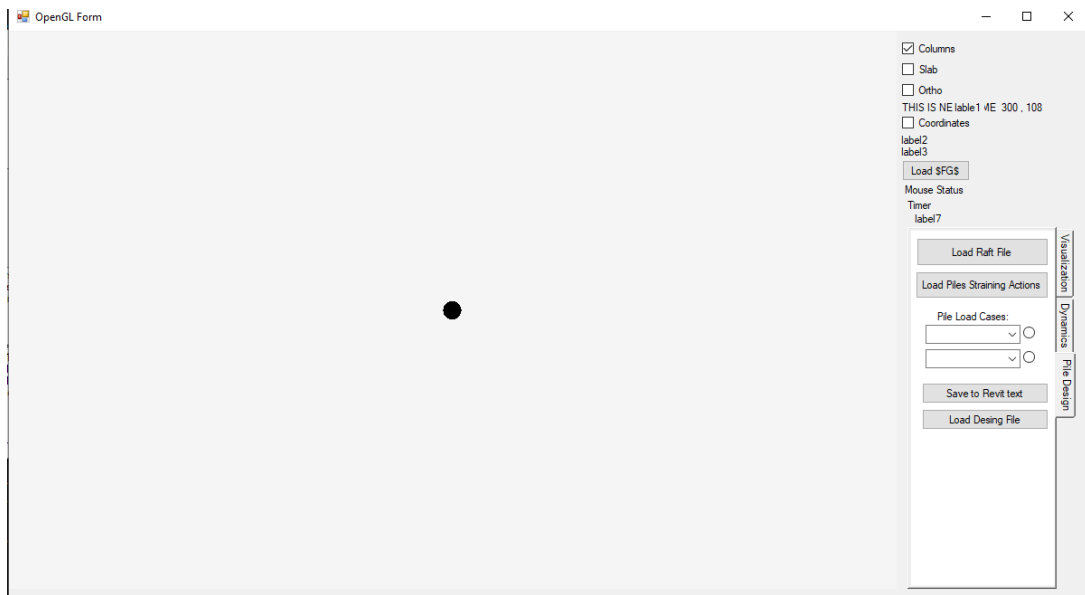


Figure 22: ‘File Design’ tab is Used to load raft and piles straining actions data.

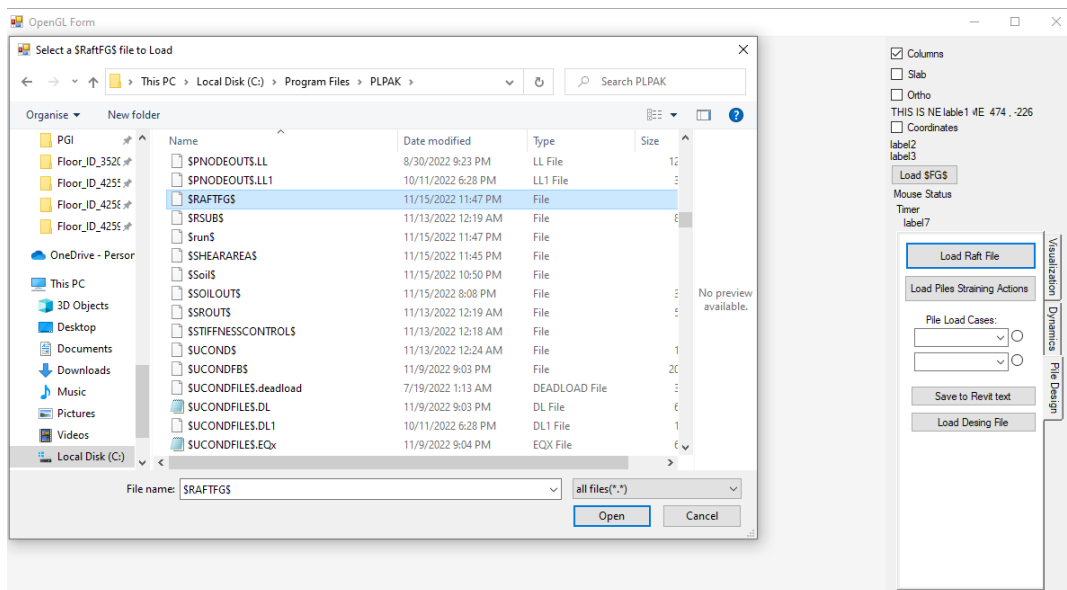


Figure 23: ‘\$RaftFG\$’ file is loaded from PLPak directory.

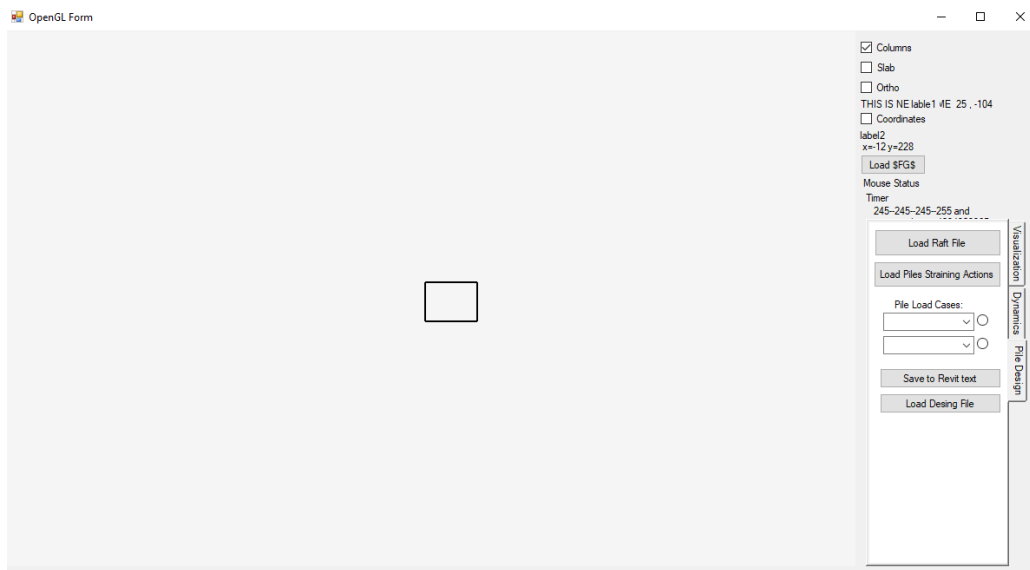


Figure 24: The View of the Raft After Loading '\$RaftFG\$', No Piles Appeared Yet.

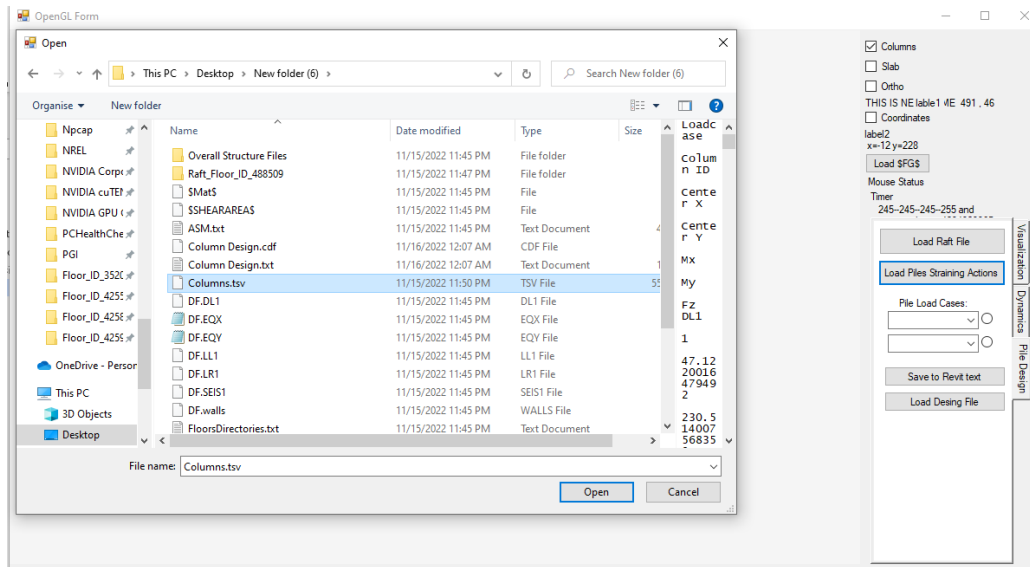


Figure 25: Loading Piles Straining Action 'Column.tsv' file from model directory.

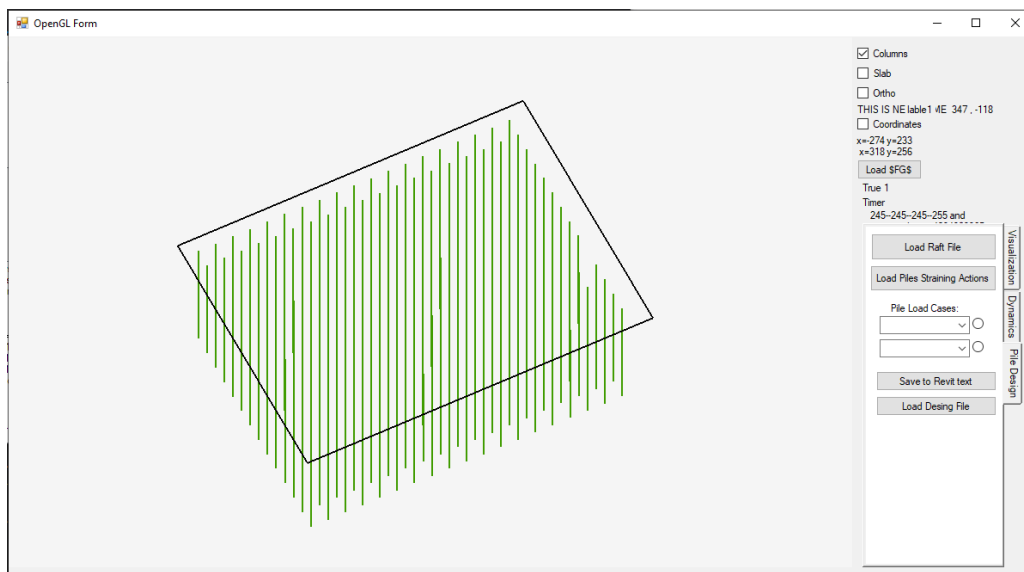


Figure 26: The View of Piles After Loading ‘Columns.tsv’.

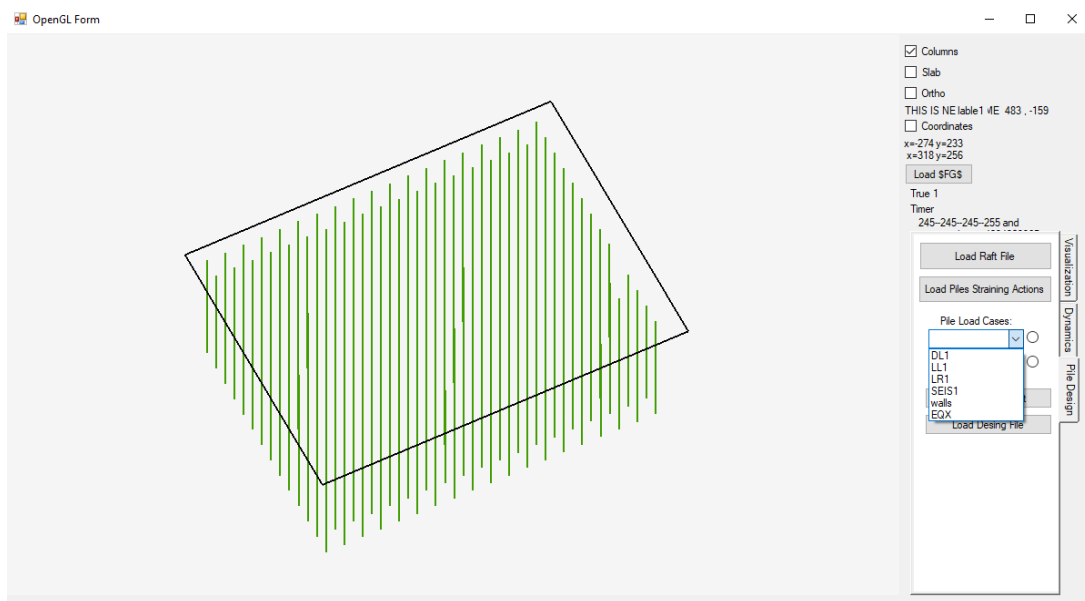


Figure 27: Picking a Loadcase to View Using Load Cases Drop Menu.

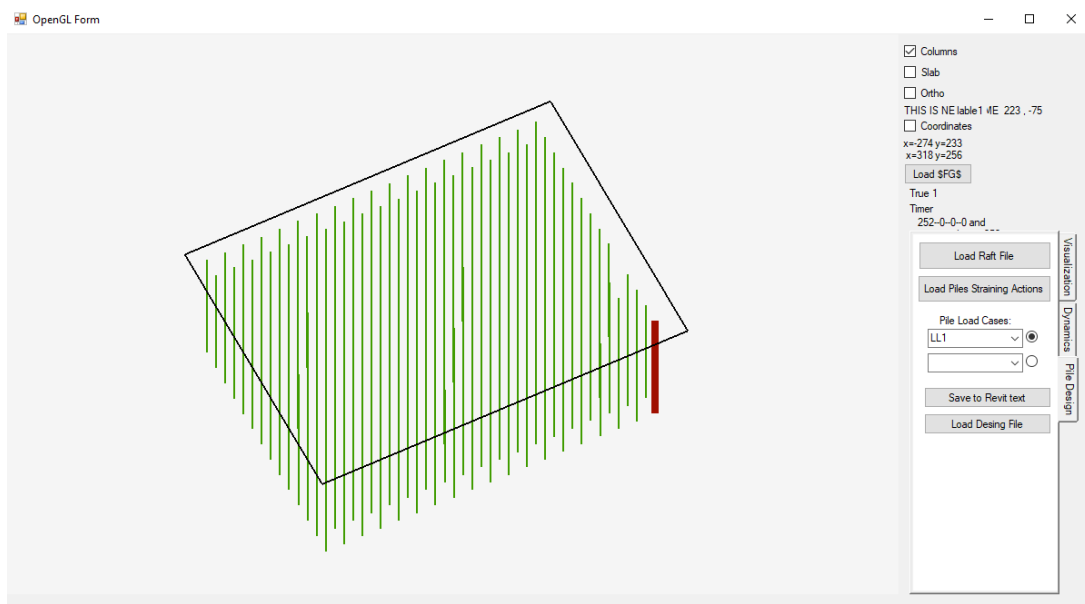


Figure 28: ‘Left Click’ Picking a Pile to Load the Pile Design Form.

Figure 29: Filling Material and Reinforcement Data Presents the Factor of Safety Against Applied Straining Actions.

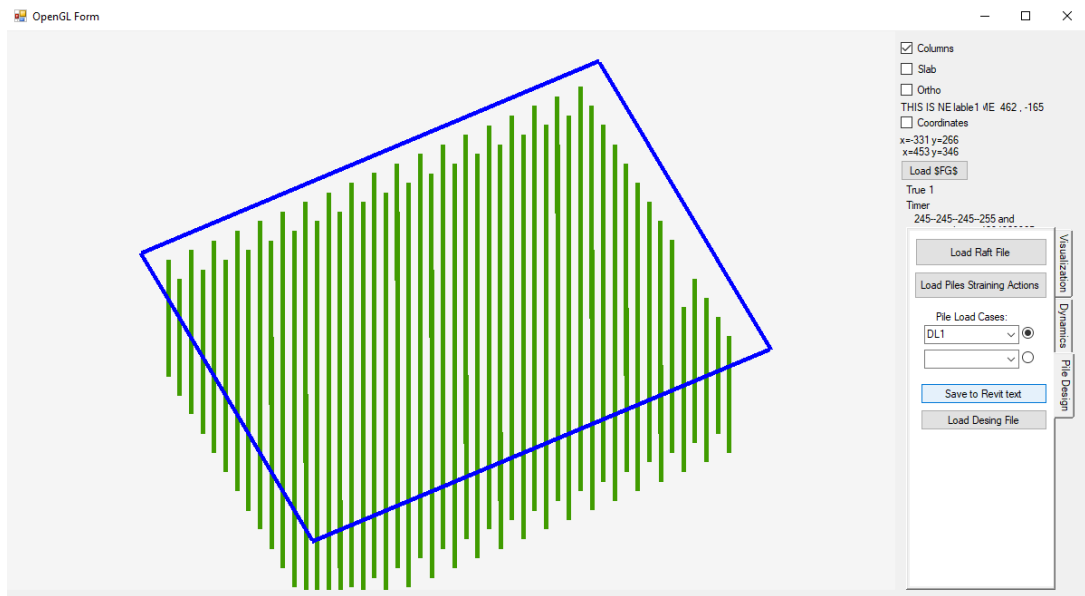


Figure 30: Export Reinforcement to Revit Using ‘Save to Revit text’.

- **Dynamic results presentation**

For showing the dynamic results as a deformed structure, the user loads the '\$FG\$' file then changes the viewing style to Wireframe style as shown in Figures 31 and 32, then the user changes to the dynamic tab.

There are two modes to view the dynamic result the first mode is to view the modal analysis result and the second is to view the time history analysis results. To view the time history analysis, the user must select the time history analysis result from the 'dynamics' folder for the model as shown in Figure 33 then the user must choose a scale to view the time history results with the scaling tool presented in Figure 34 after that the user can switch between the time steps using the times step switch as presented in Figure 35, for the unloading of the result the user clicks the same button 'Unload Time History'.

To view the modal results, the user loads the mood shape file 'mode' from the 'dynamics' directory for the problem as presented in Figure 36. Subsequently, the user uses the same mode order picking tool to select the mode shape order and view the results as presented in Figure 37.

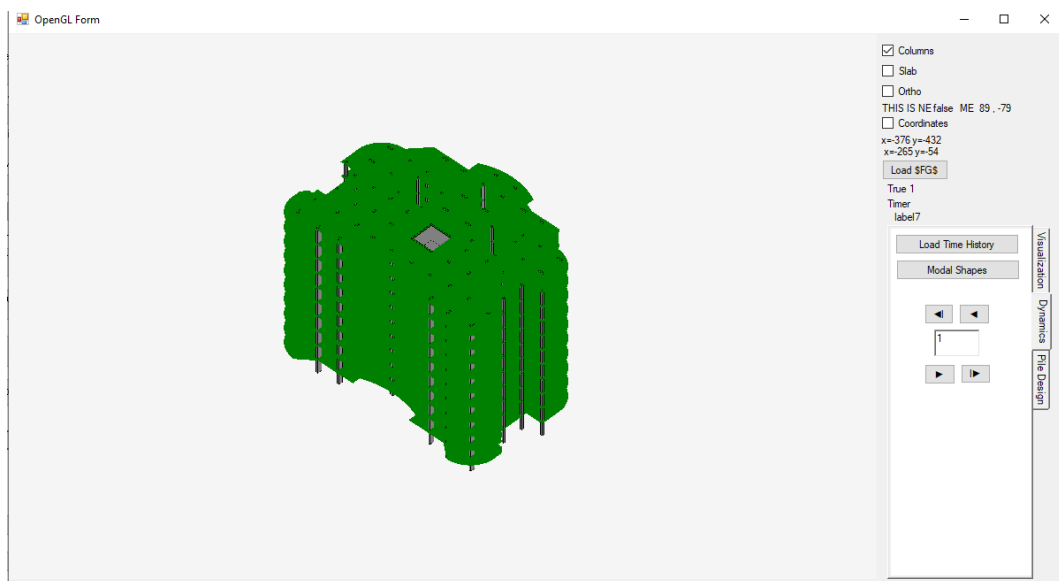


Figure 31: The Extruded View for \$FG\$ Dynamic Results.

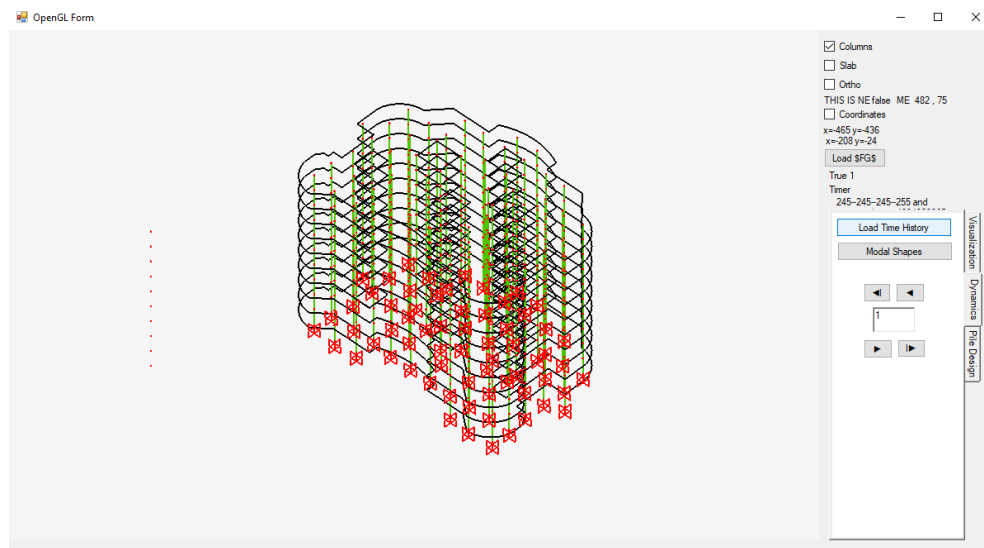


Figure 32: The Wireframe View for SFGS Dynamic Results.

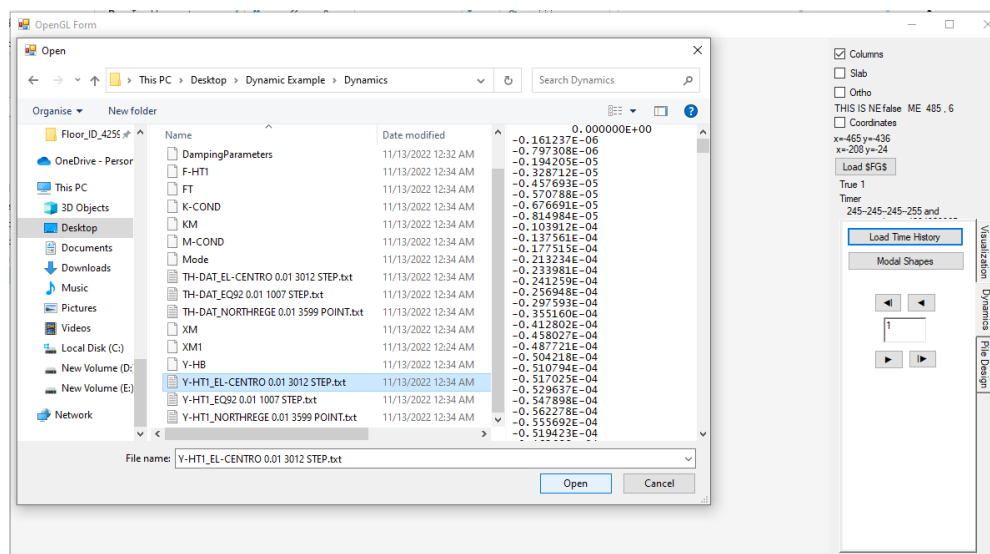


Figure 33: Loading the Time History Results.

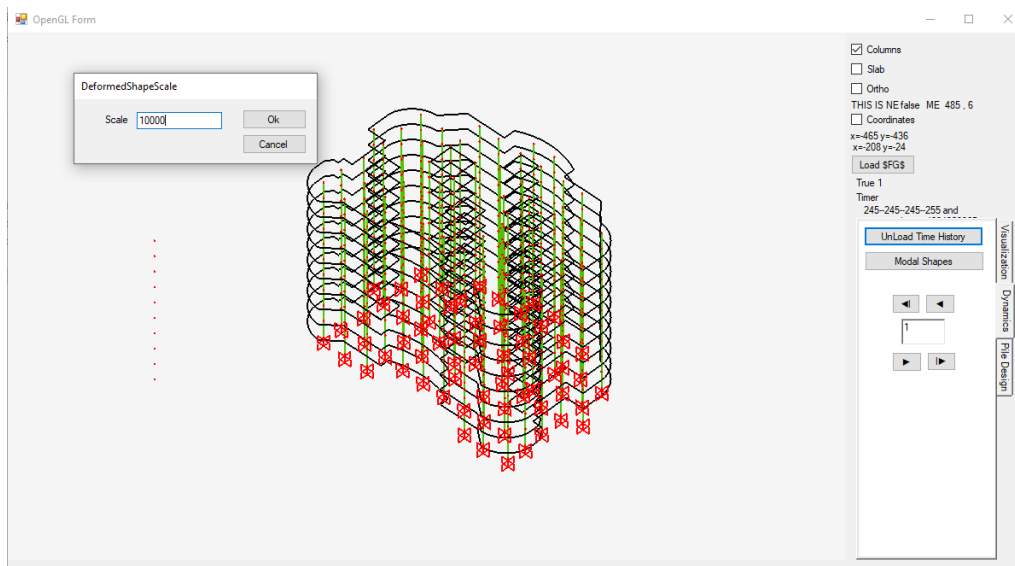


Figure 34: Choosing the Scale for Time History Results.

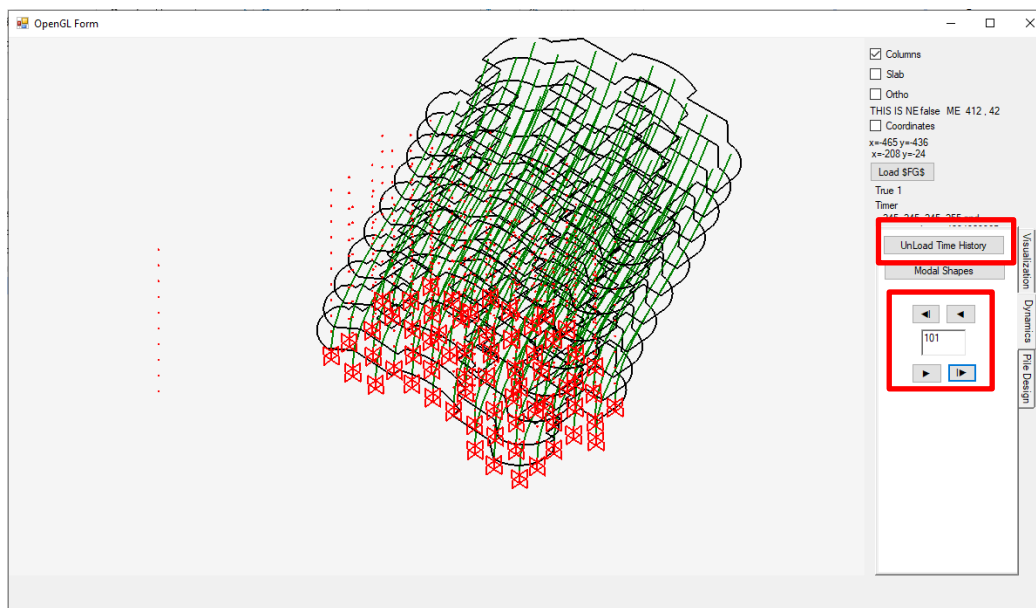


Figure 35: The Deformed Results of Step 101, 'Unload Time History' button unloads the results.

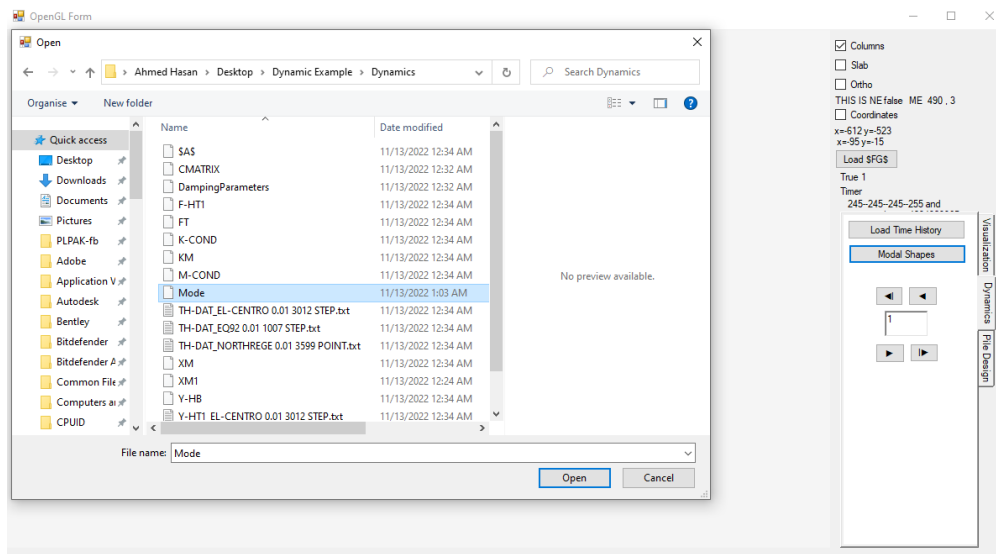


Figure 36: Choosing the 'Mode' File for Modal Analysis Results.

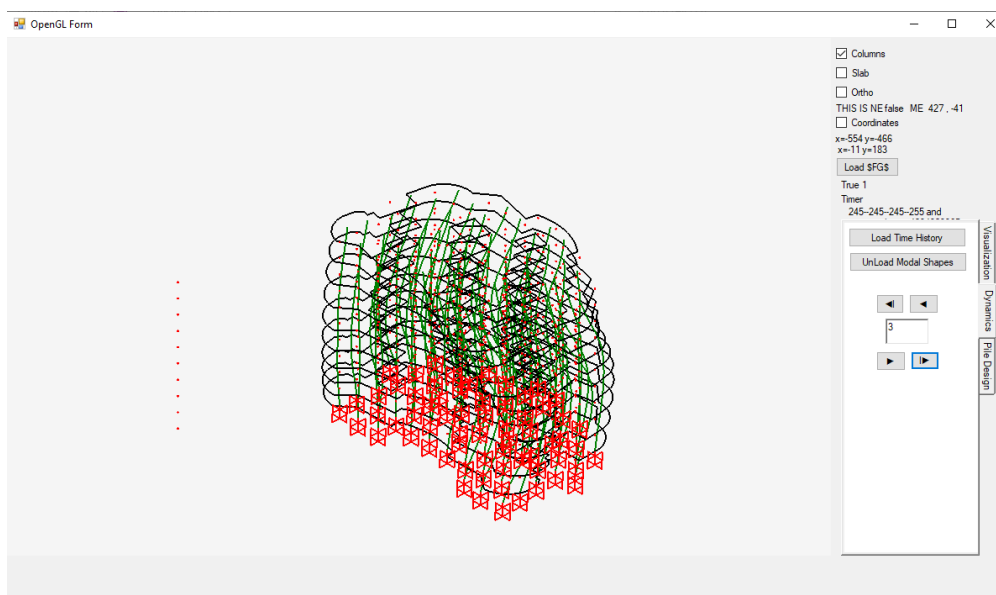


Figure 37: The Deformed Result for Mode Shape 3.

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