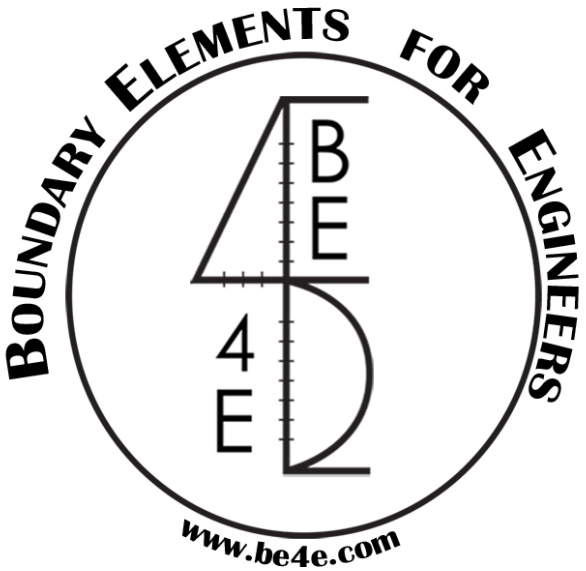


The Post-Tension Tool

User MANUAL

PLPAK™ Version 2.00

STRUCTURAL ANALYSIS SOFTWARE USING

THE BOUNDARY ELEMENTS METHOD

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

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:

PTPAK (Post tension package) is a structural design tool package for pre-stressed (post-tensioned or pre-tensioned) plate bending structures based on the boundary element method for shear deformable plate bending theory, using different codes like (ACI, EC, and ECP).

The PTPAK is added to the PLPAK-Basic package to design reinforced concrete building slabs and foundations.

The PTPAK is not only consider about design, but also about detailing and calculation sheet forming.

In PTPAK the user can change the cable profile (13 templates) to serve the different structure conditions.

In PTPAK the user can draw reinforcement on slab and determine the reinforcement at beams (without post tension).

In the PTPAK the calculation is not only for load combination, but also for envelopes.

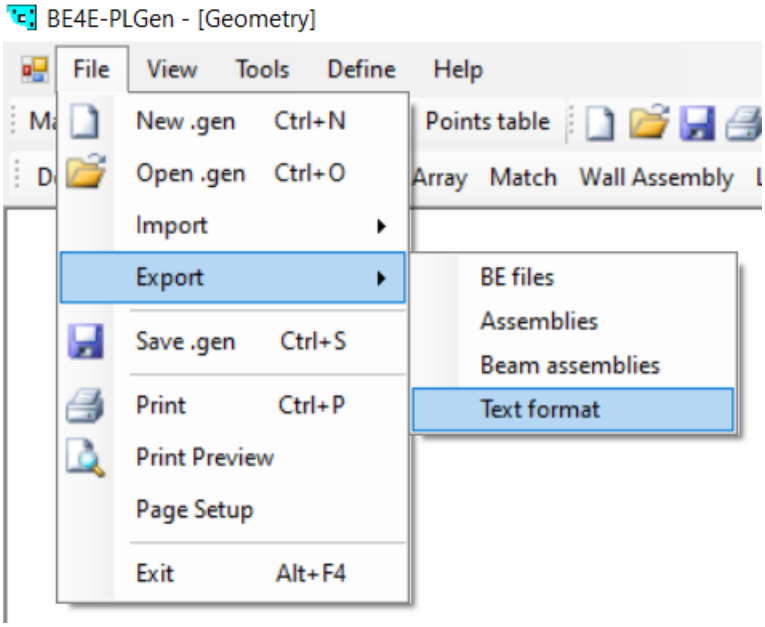
In this manual, the user can start his model from Autodesk Revit or from PLGen until finishing his model reinforcement details in Autodesk Revit.

Therefore, this manual will explain the following:

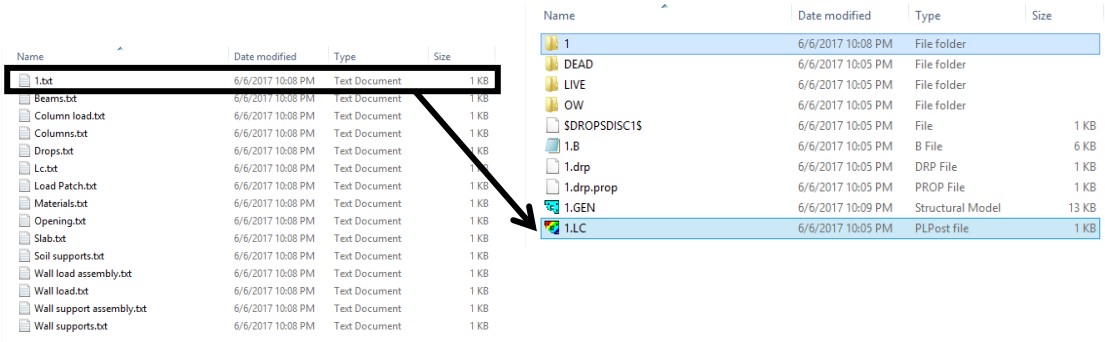
1. Loading already analyzed slab.
2. Defining material properties.
3. Defining losses parameters.
4. Drawing cables.
5. Profiling cables.
6. Solving PT load cases.
7. Drawing PT design strips.
8. Solving the strips.
9. Checking stresses result.
10. Exporting cables to Autodesk Revit.
11. Optimization tool.

1. Loading already analyzed slab

- After drawing the problem (without post tension) in PLGen as shown below BE-Files should exported and the text files also for the problem.

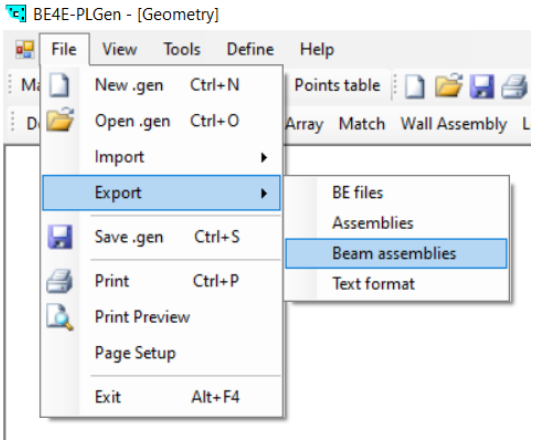


- Note that when exporting Text format files, enter the same name of the .LC file in the "File name:" text box of the exporting window, and exported them to a folder with the same name and path of the .LC file.

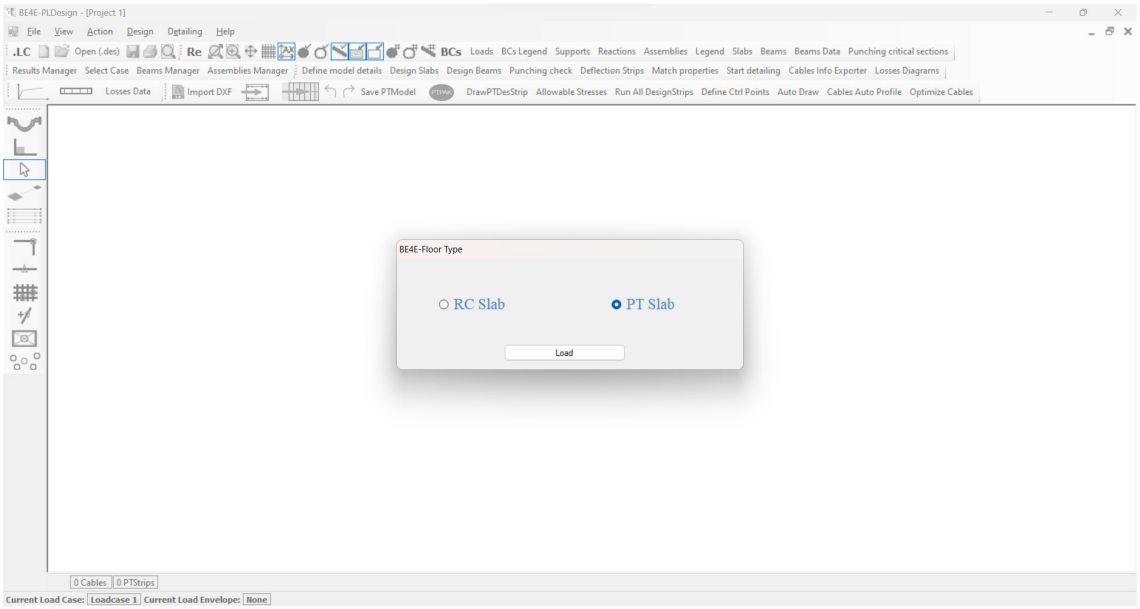


-

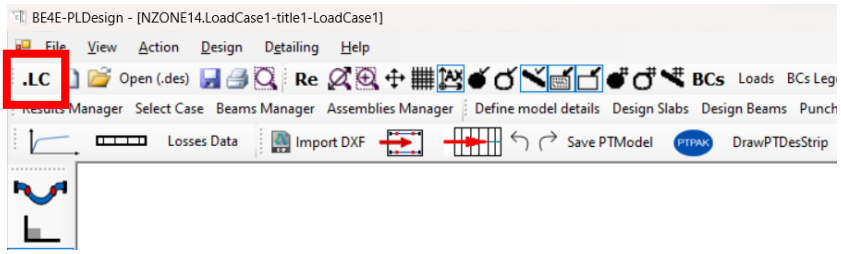
- Beam assemblies should be exported if there are beams.

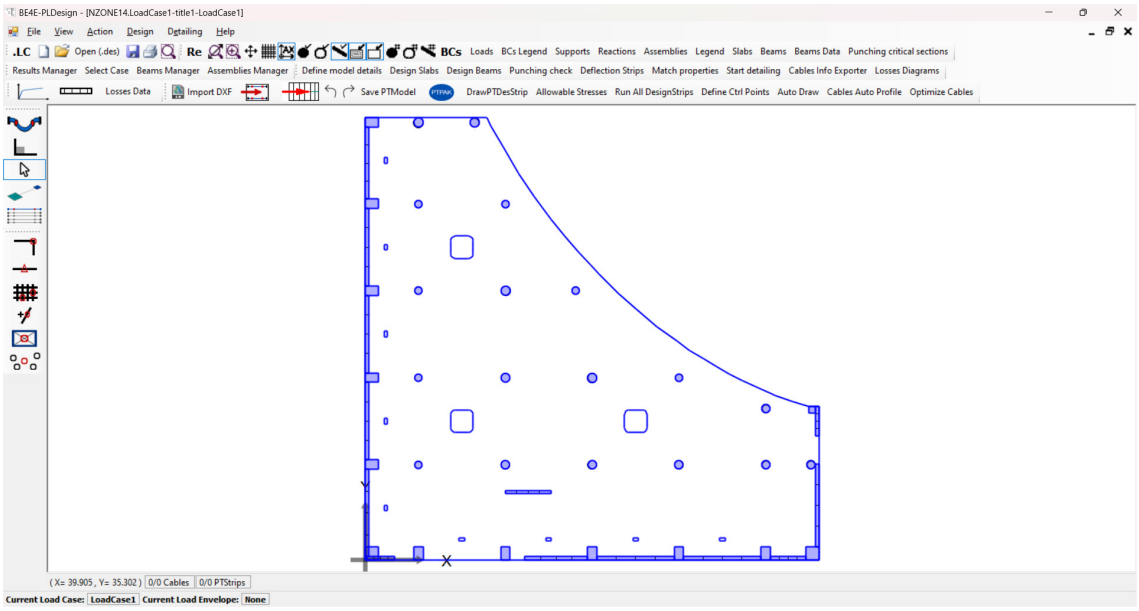


- First the user should open the PLDesign tool and choose the “PT Slab” option as shown below.



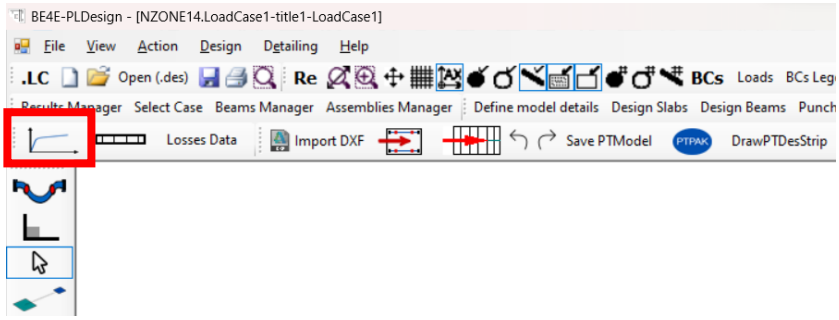
- Load the .LC file of the problem to load the model.



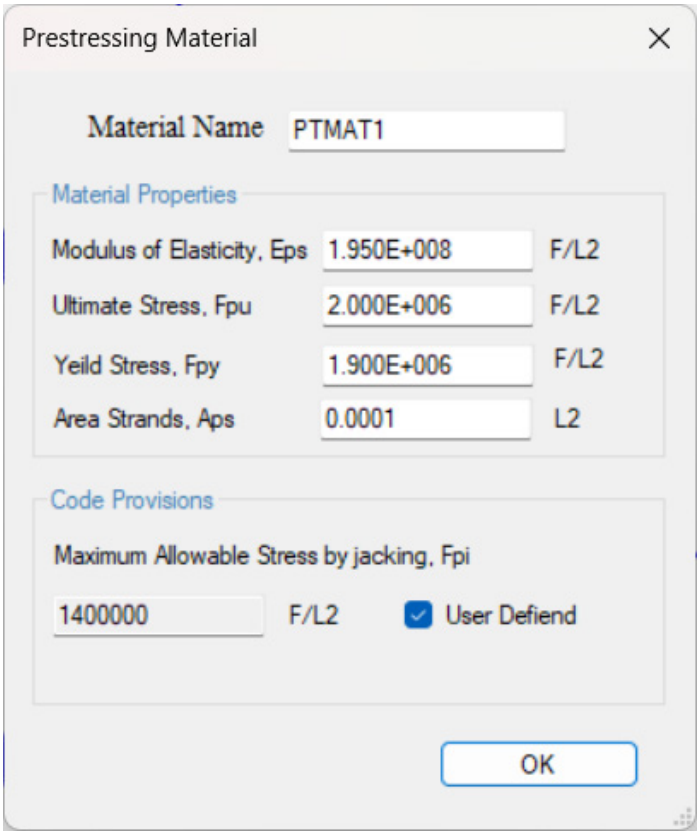


2. Defining material properties

- Open the prestressing material properties window.

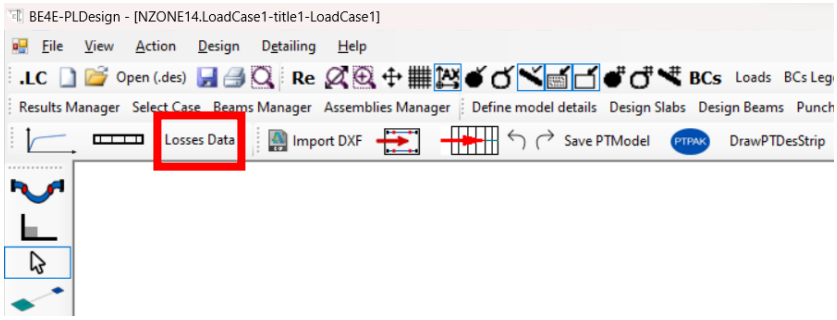


- Hence, define the material properties.

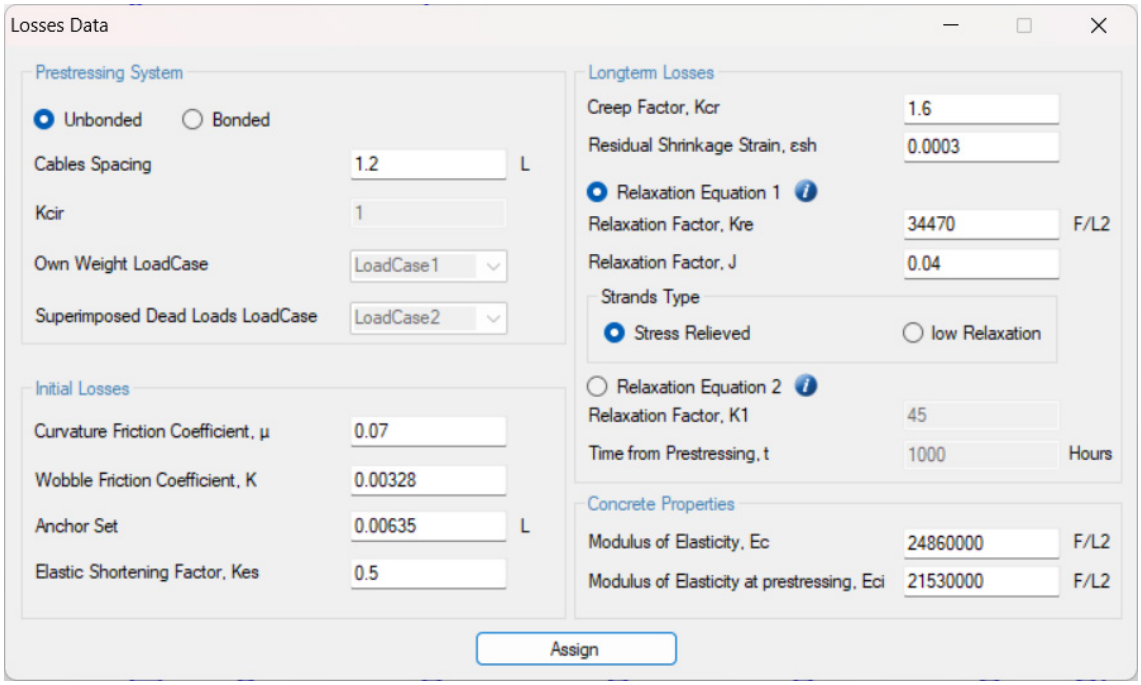


3. Defining losses parameters

- Open the losses data window.



- Enter the losses parameters based on the required design code.

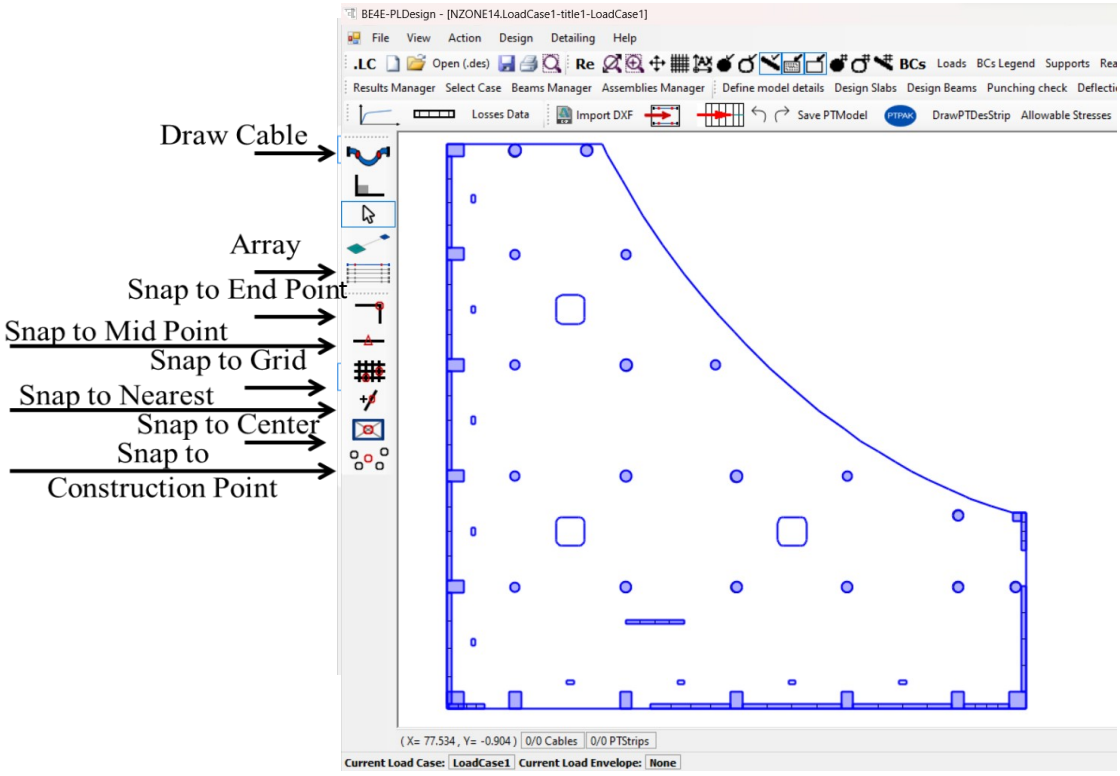


4. Drawing cables

The user can draw cables using 3 ways manual drawing, importing from DXF file, or using the Auto Draw capability (Post-Tension Automation Tool is required for this option).

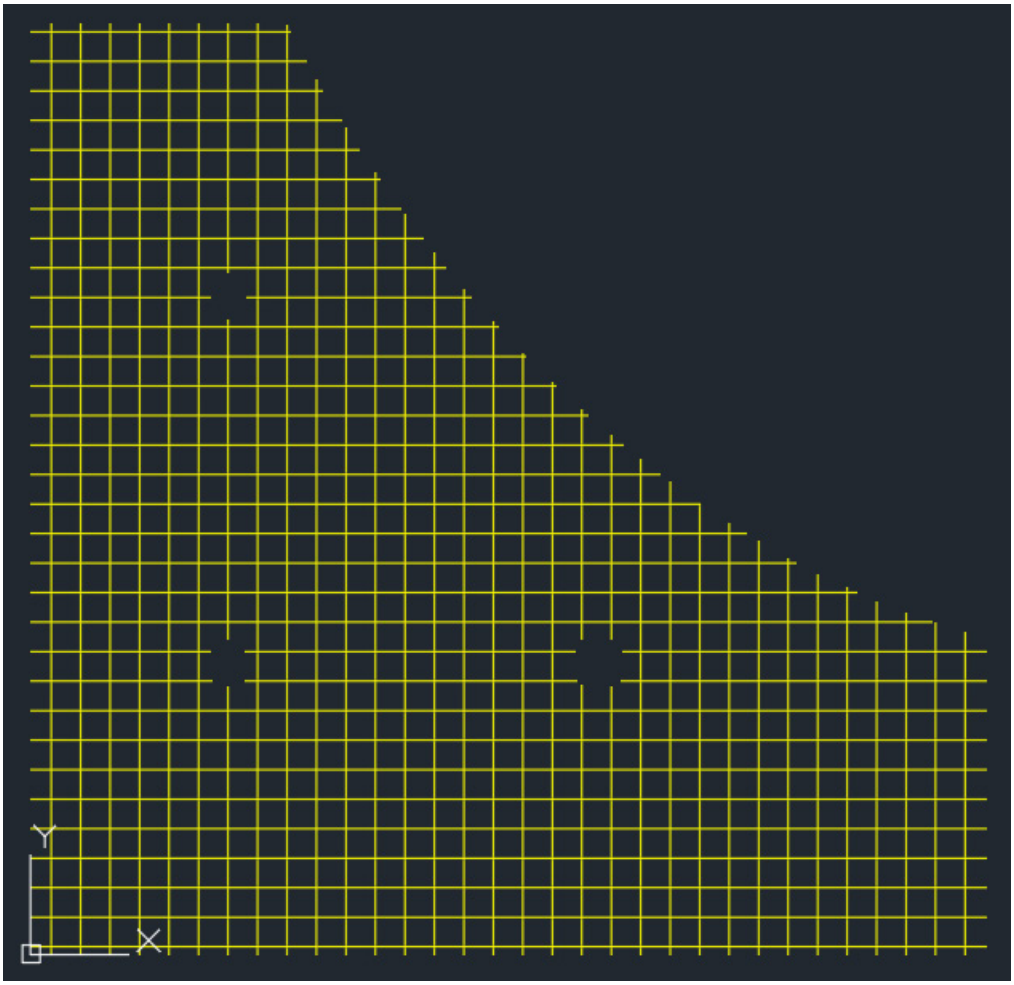
4.1. Manual drawing

The user can draw the cables manually with the help of the snaping tools.

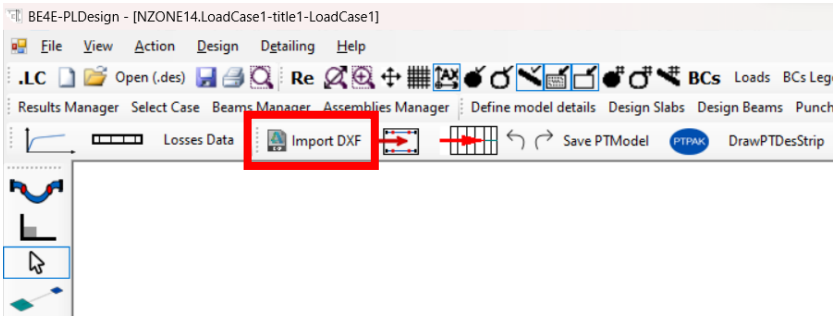


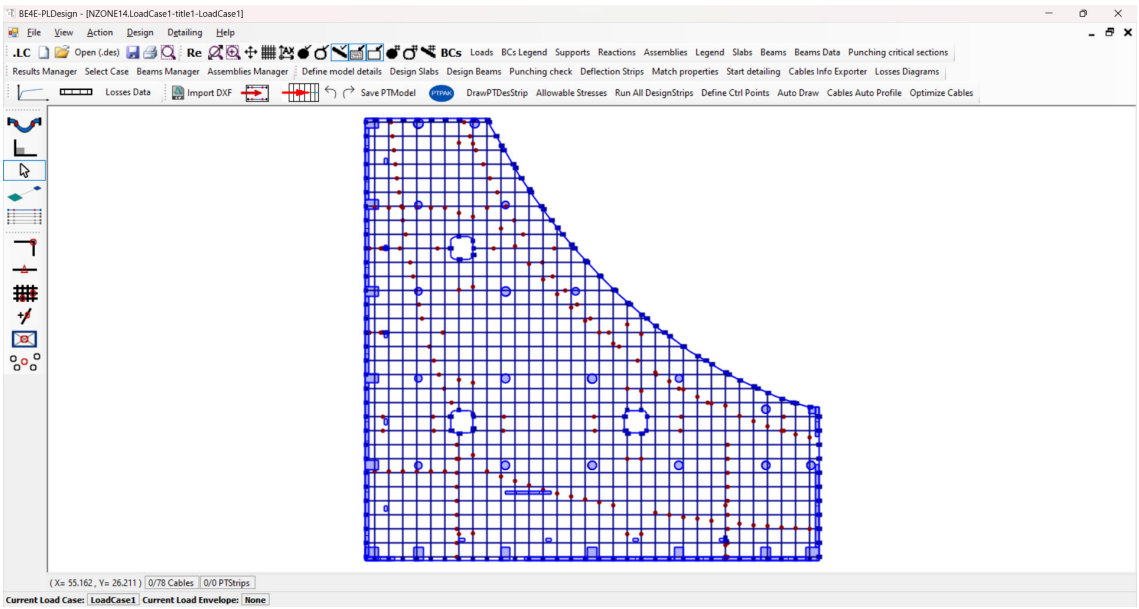
4.2. Importing from DXF file

- The user can draw cables in AutoCad as polylines. In these polylines, additional points should be added at the end of each segment. Then, the user should save the file in DXF format.



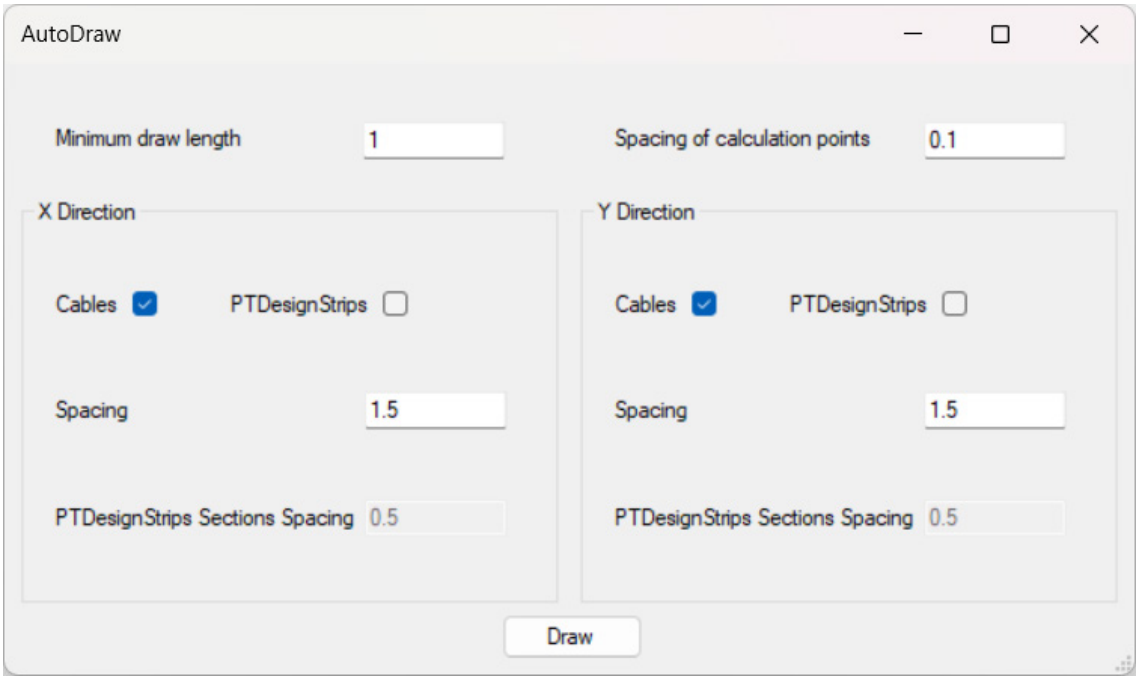
- Hence, the user can import the saved DXF file by clicking on Import DXF button.

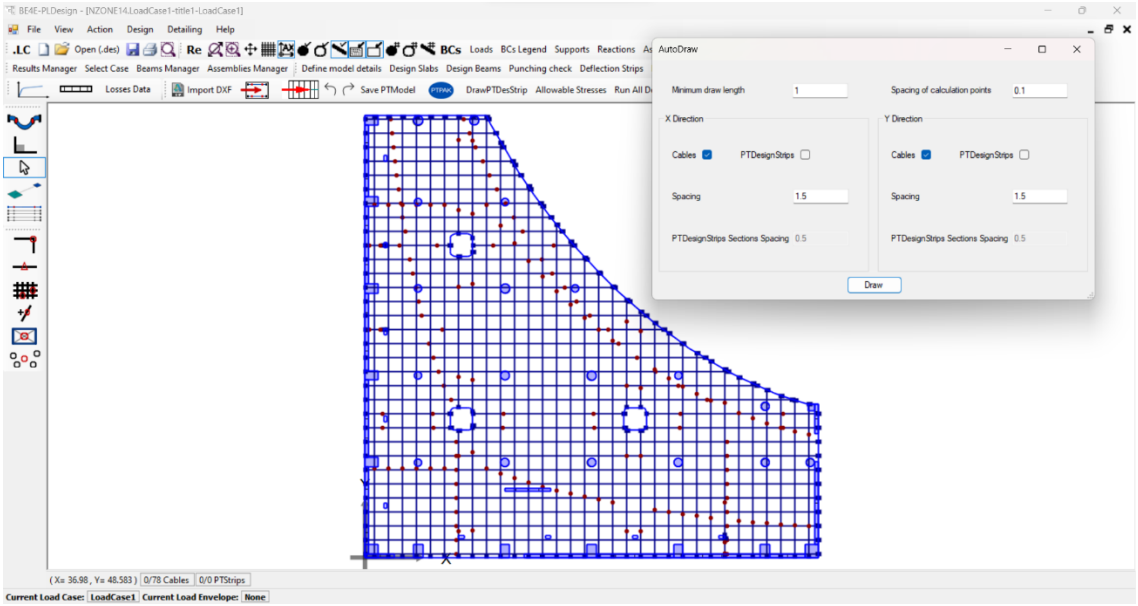




4.3. Auto Draw capability (Post-Tension Automation Tool is required for this option)

The user can draw cables automatically using the Auto Draw capability by choosing to draw cables in x direction and/or cables in y direction. The spacing of these cables should be specified and also a minimum draw length is required to ignore cables with small lengths. The spacing of calculation points is the tolerance that determines the maximum distance between cable start/end point and the slab edges. It should be small and bigger than zero.



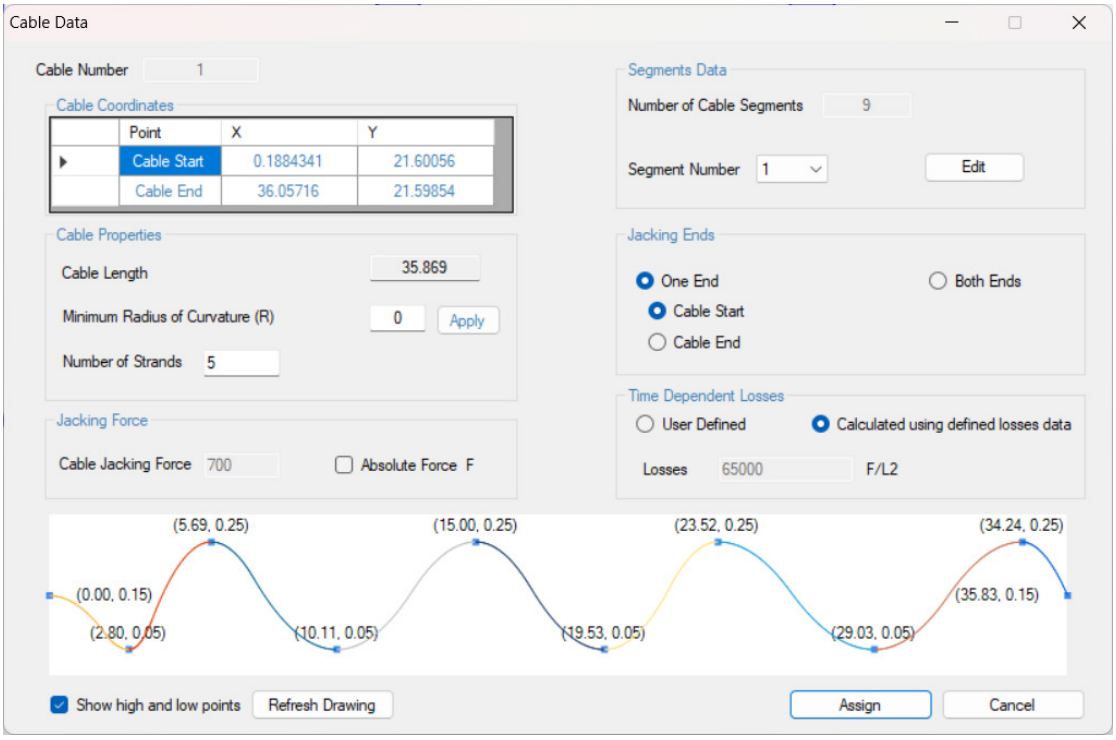


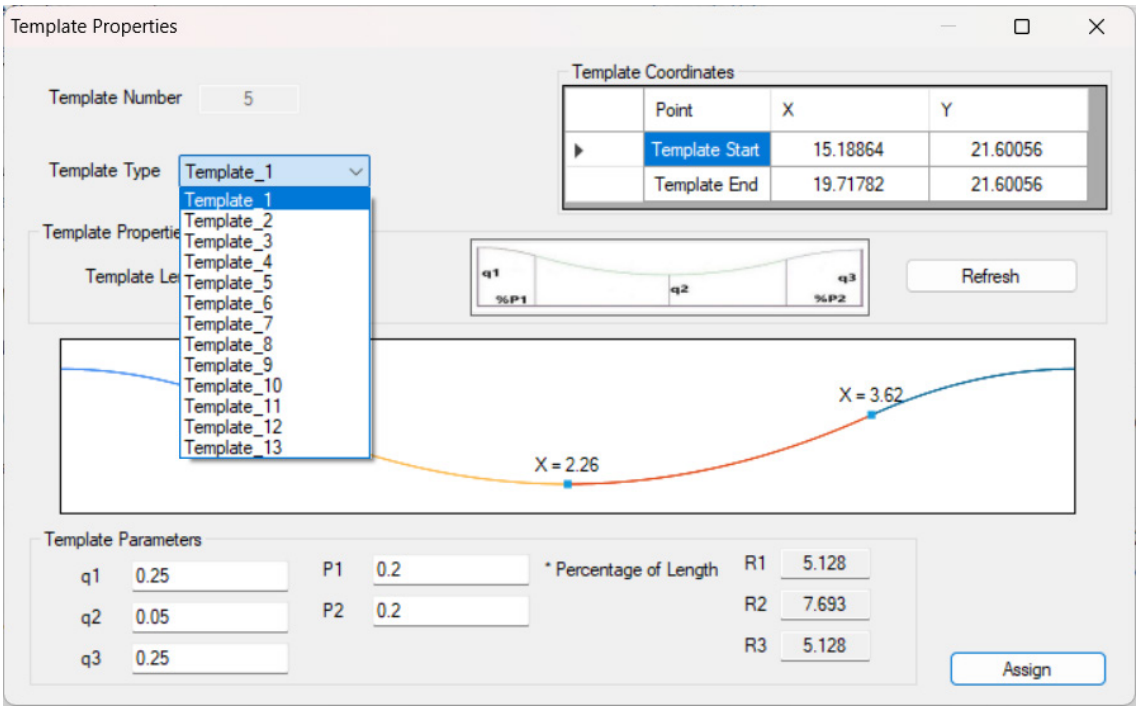
5. Profiling cables

The user can profile cables using 2 ways, manual profiling, or using the Auto Profile capability (Post-Tension Automation Tool is required for this option).

5.1. Manual profiling

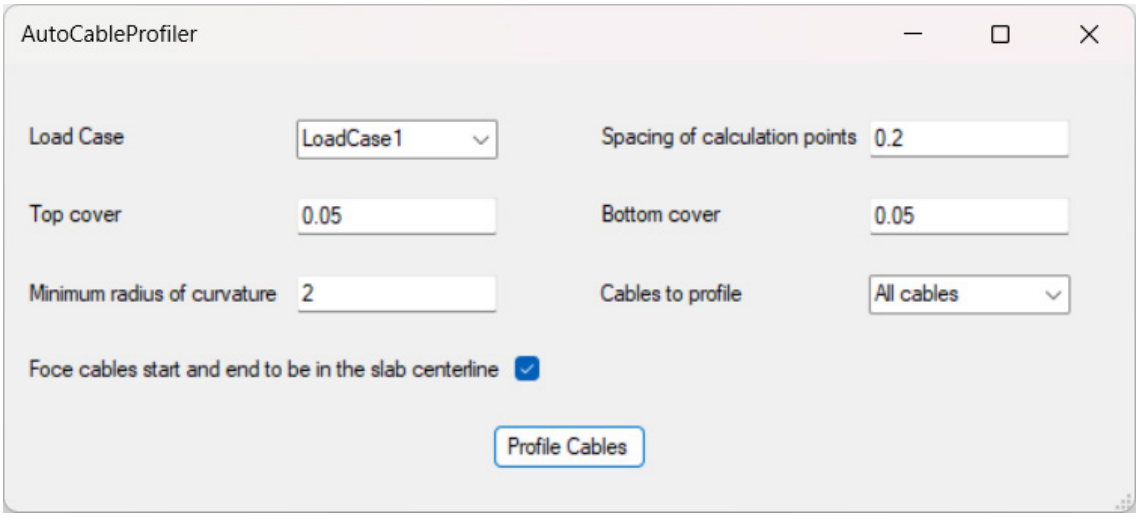
The user start profiling the desired cable by right clicking on it and the following window will appear. Throw this window the user can define number of strands, jacking force and jacking ends. Also, the time dependent losses can be user defined or calculated automatically using defined parameters. Hence, the profile of each segment can be defined from 13 templates by choosing segment number and clicking on Edit.

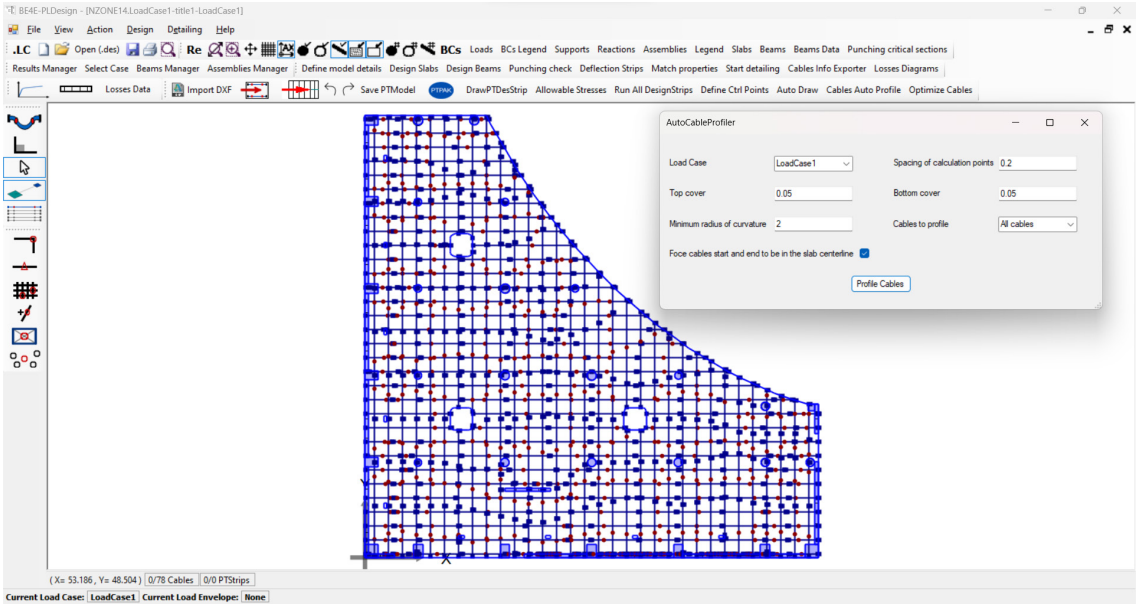




5.2. Auto Profile capability (Post-Tension Automation Tool is required for this option)

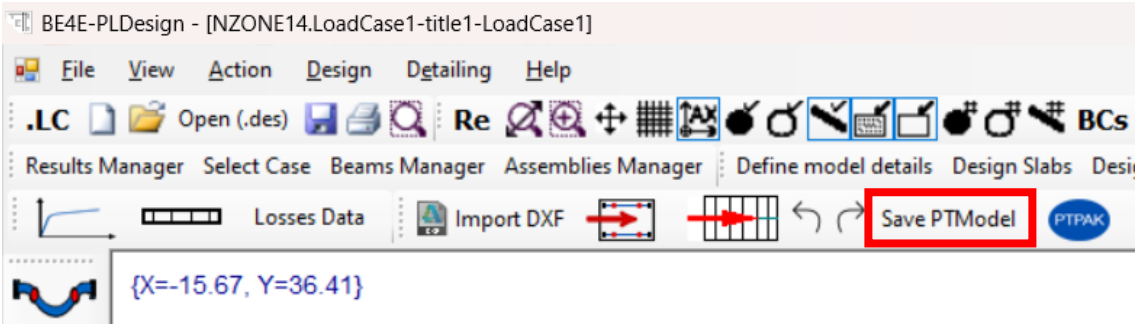
The user can profile the cables automatically using the Auto Profile capability by choosing which load case to profile on, the spacing of the points used to calculate deflection, the top and bottom slab covers, the minimum radius of curvature, the cables to be profiled (selected cables or all cables), and finally whether he wants the start and the end of the cable to be in the slab centerline or not.



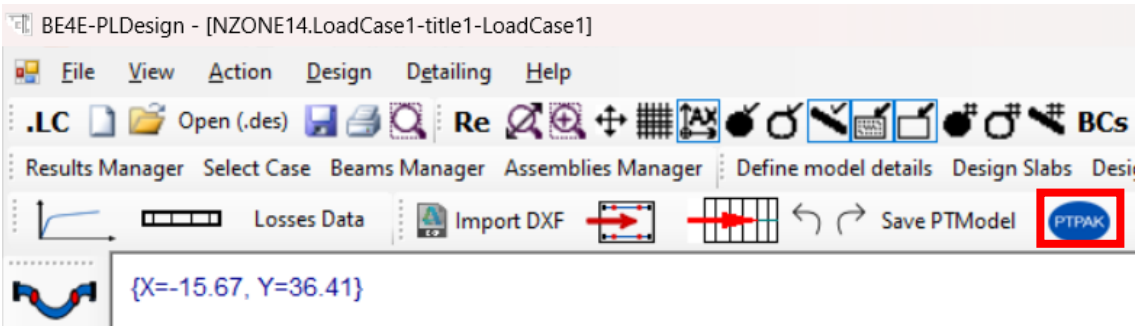


6. Solving PT load cases

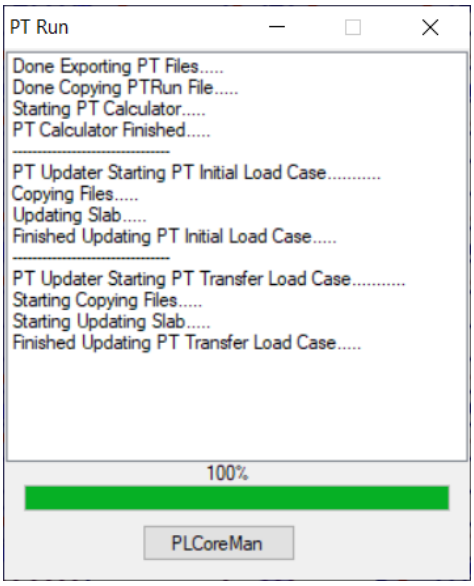
- The user should save the PTModel.

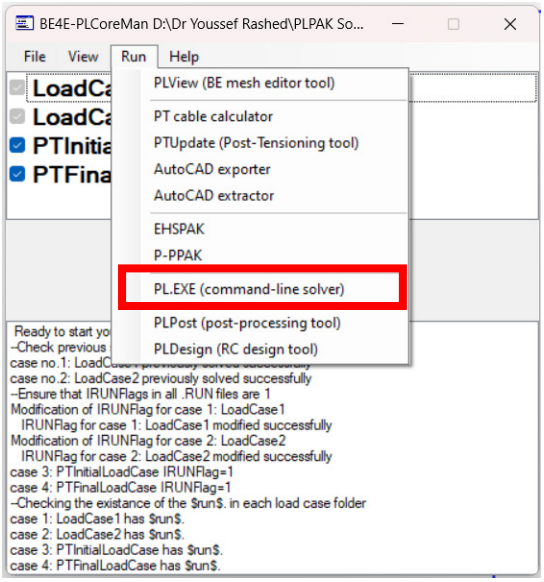


- Hence, click on the PTPAK button.



- Click PLCoreMan, and run PL.exe



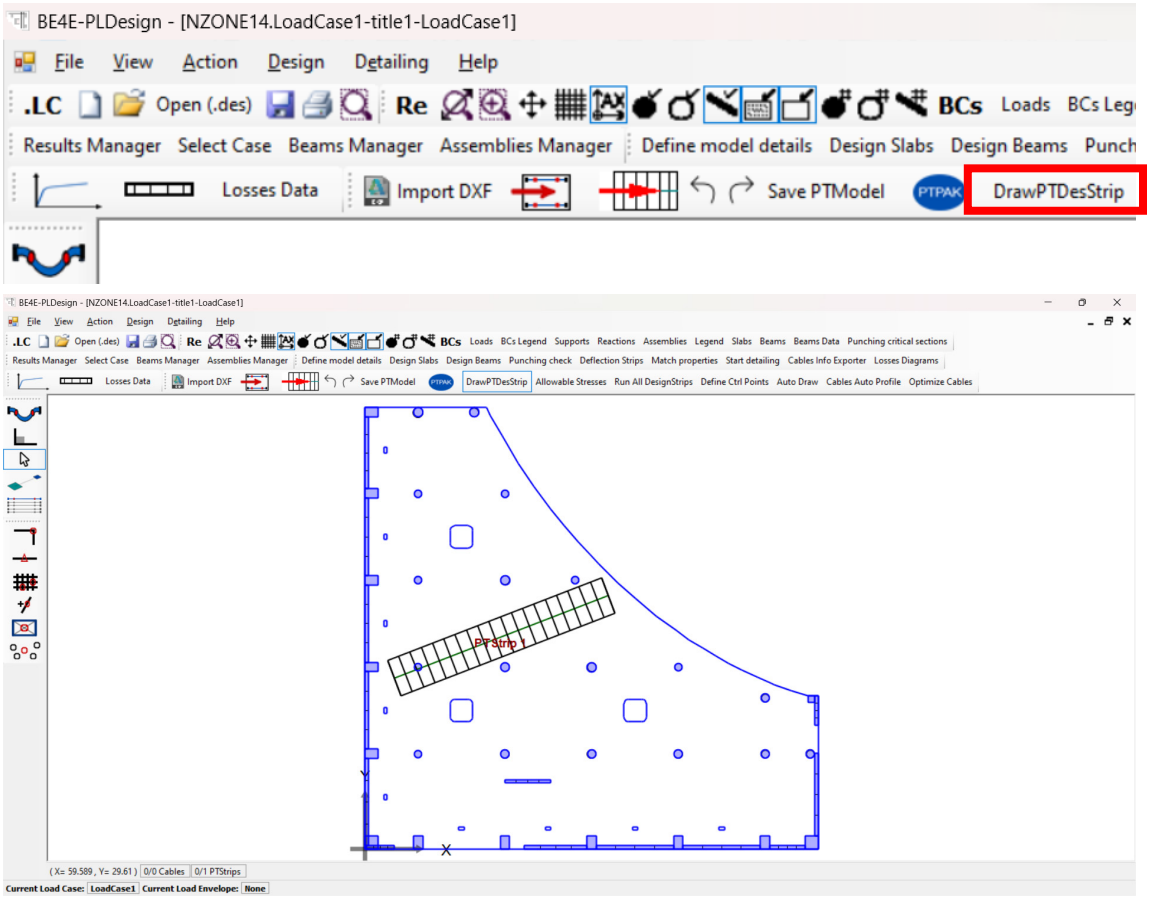


7. Drawing PT design strips

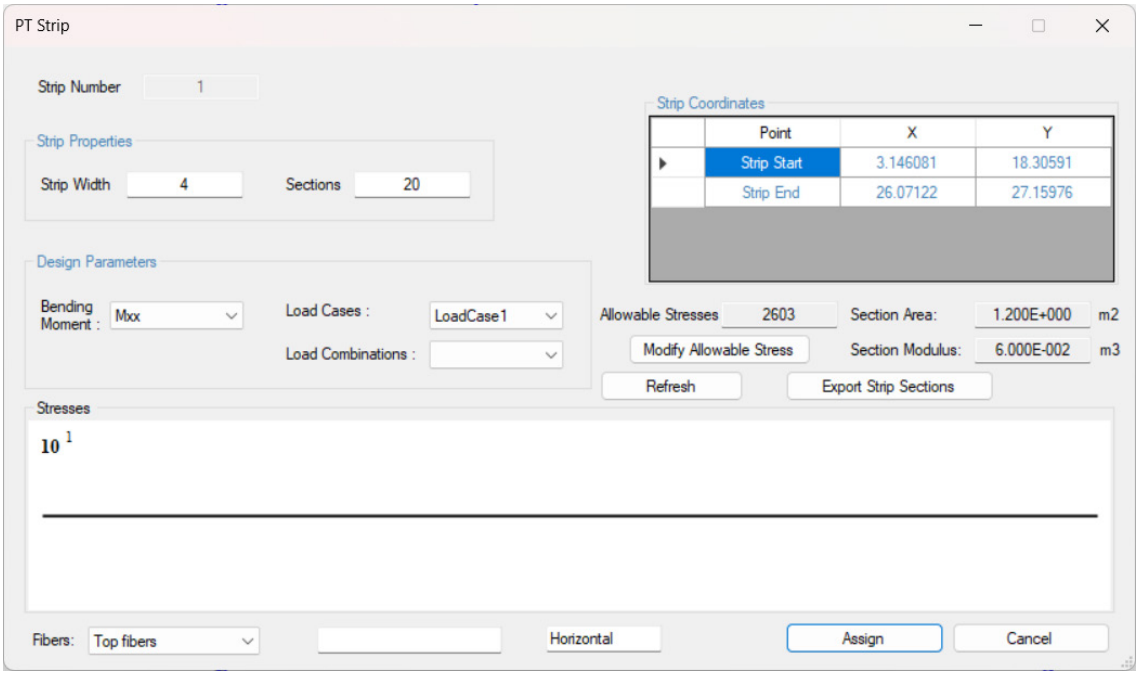
The user can draw PT design strips using 2 ways, manual drawing, or using the Auto Draw capability (Post-Tension Automation Tool is required for this option).

7.1. Manual drawing

- The user can draw the PT design strips manually by clicking on DrawPTDesStrip button and clicking the start and end points.

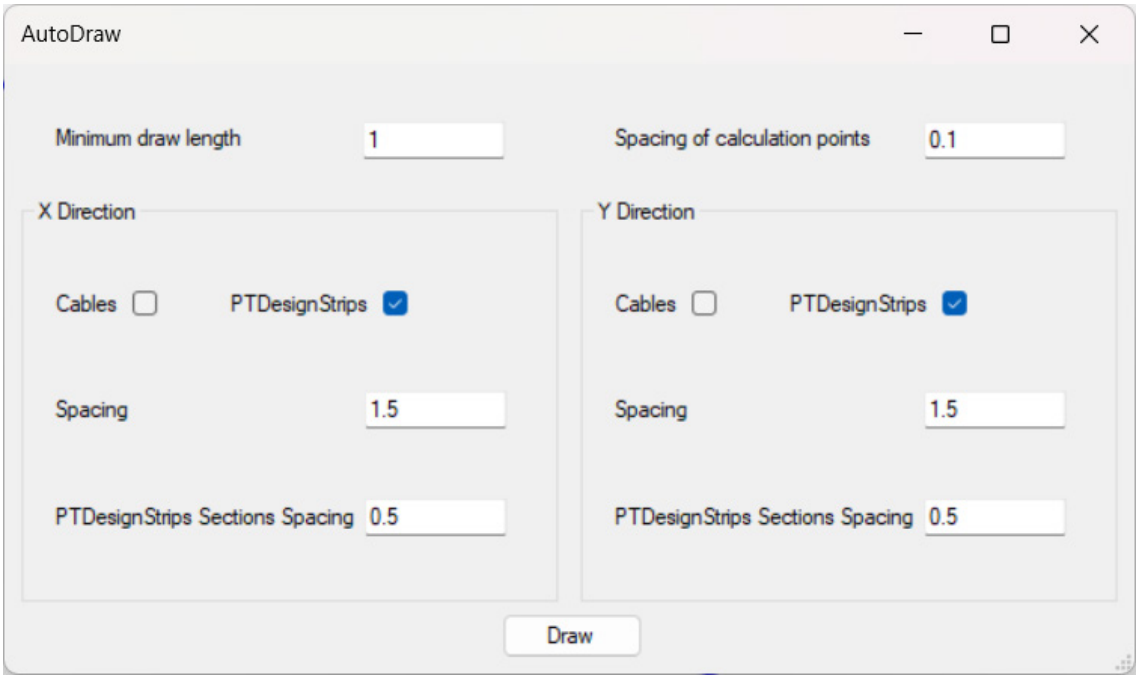


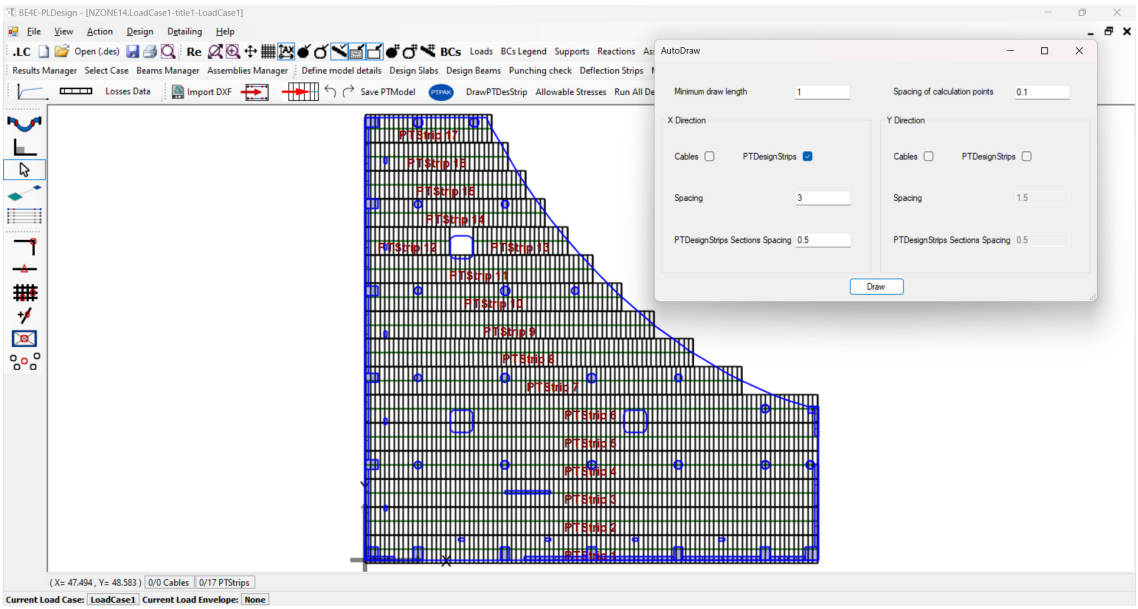
- Right click on the strip centerline to modify the strip width and number of sections.



7.2. Auto Draw capability (Post-Tension Automation Tool is required for this option)

The user can draw the PT design strips automatically using the Auto Draw capability by choosing to draw strips in x direction and/or strips in y direction. The spacing of these strips should be specified and also a minimum draw length is required to ignore strips with small lengths. The spacing of calculation points is the tolerance that determines the maximum distance between strip start/end point and the slab edges. It should be small and bigger than zero.



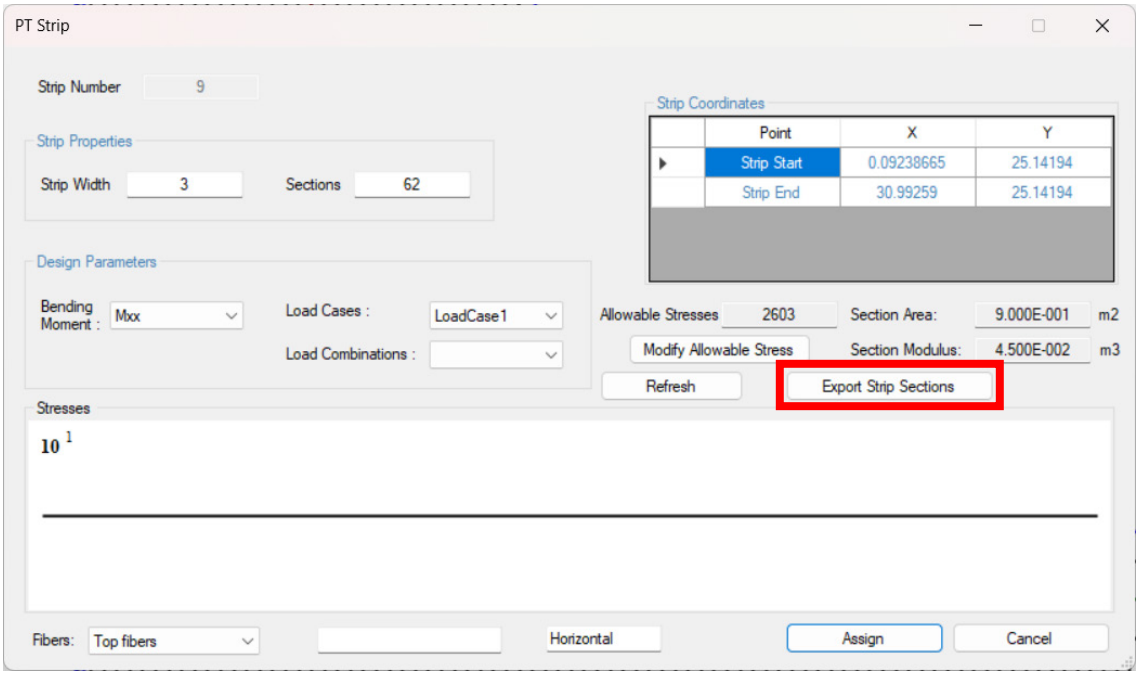


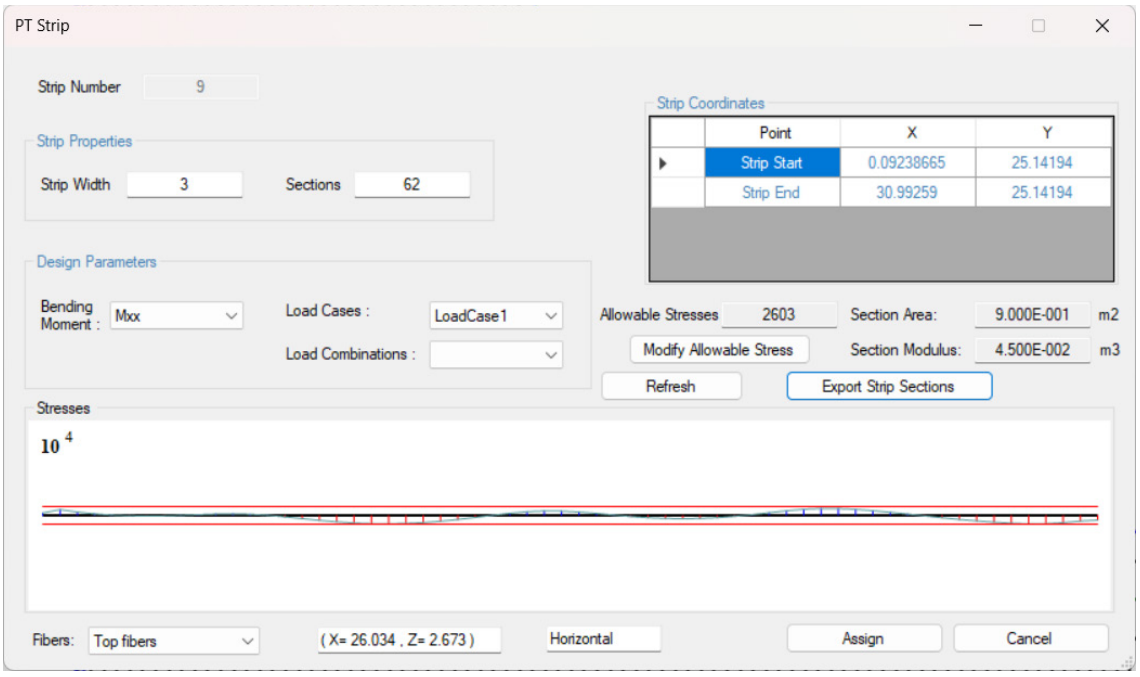
8. Solving the strips

The user can solve PT design strips using 2 ways, solving specific strip, or all design strips.

8.1. Specific strip

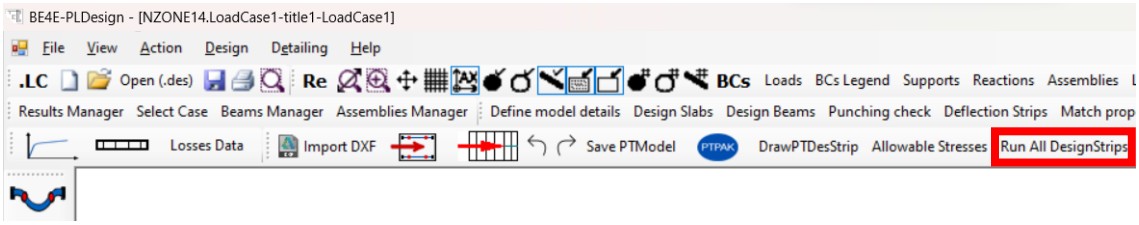
- The user can solve a specific PT design strip by right clicking on the strip centerline. Hence, click on Export Strip Sections button.





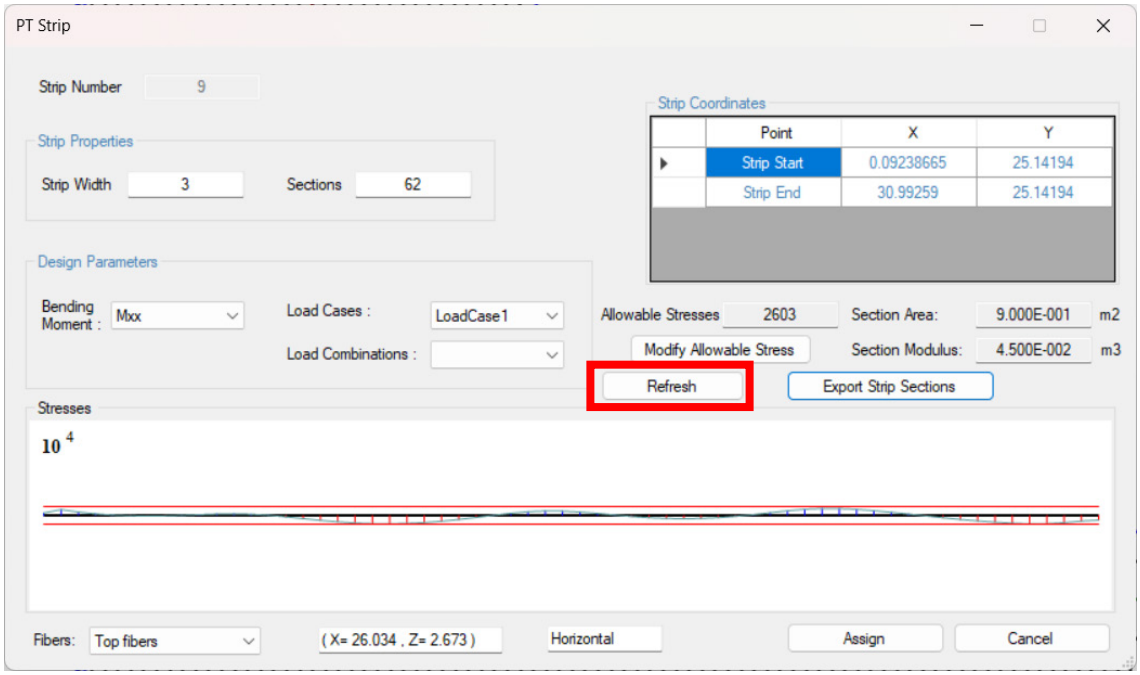
8.2. All design strips

The user can solve all PT design strips automatically by clicking on Run All DesignStrips button.



9. Checking stresses result

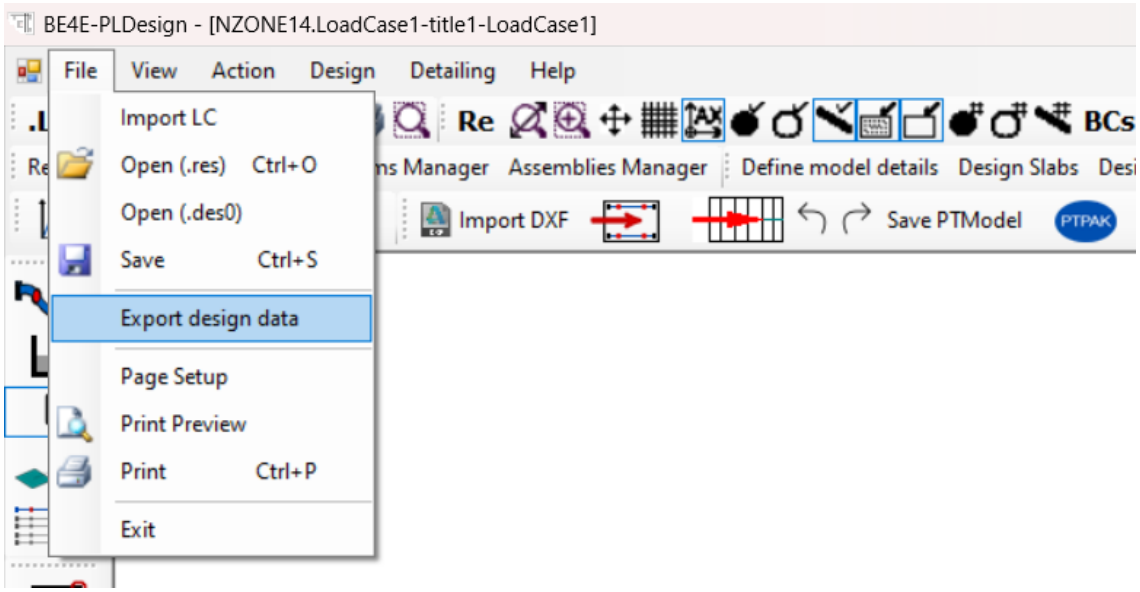
The stress results for the PT design strips can be demonstrated by right clicking on the centerline of the strip on clicking refresh. The allowable stresses are defined. If the user used the Run All DesignStrips button, results are exported in a text file called $sectionsstress$ in the same path of the .LC file.



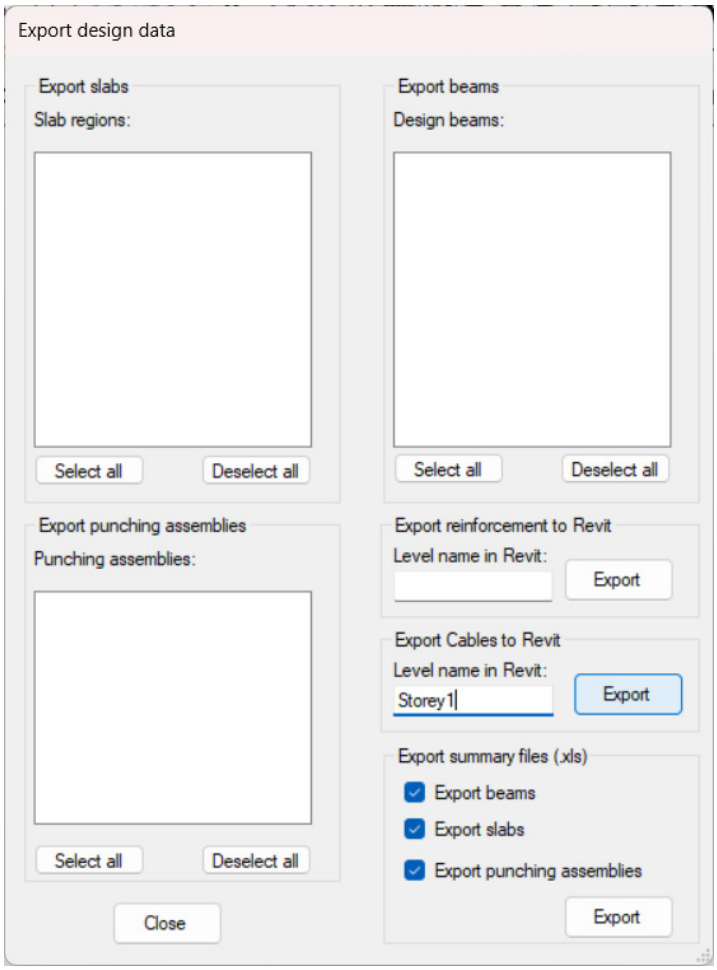

```
15
13
0
40
LoadCase1
952.02681,-952.02681
635.40772,-635.40772
-233.32687,233.32687
-95.21135,95.21135
1053.29611,-1053.29611
1878.20951,-1878.20951
-617.27338,617.27338
-2108.99193,2108.99193
-2632.66419,2632.66419
-2214.55424,2214.55424
-882.65591,882.65591
1510.4182,-1510.4182
4060.70847,-4060.70847
864.44758,-864.44758
-225.33056,225.33056
-625.00295,625.00295
-249.60741,249.60741
-726.72433,726.72433
-61.68848,61.68848
1657.57347,-1657.57347
2406.30492,-2406.30492
-62.51912,62.51912
-1550.5434,1550.5434
-2134.35871,2134.35871
-1871.37648,1871.37648
```

10.Exporting cables to Autodesk Revit

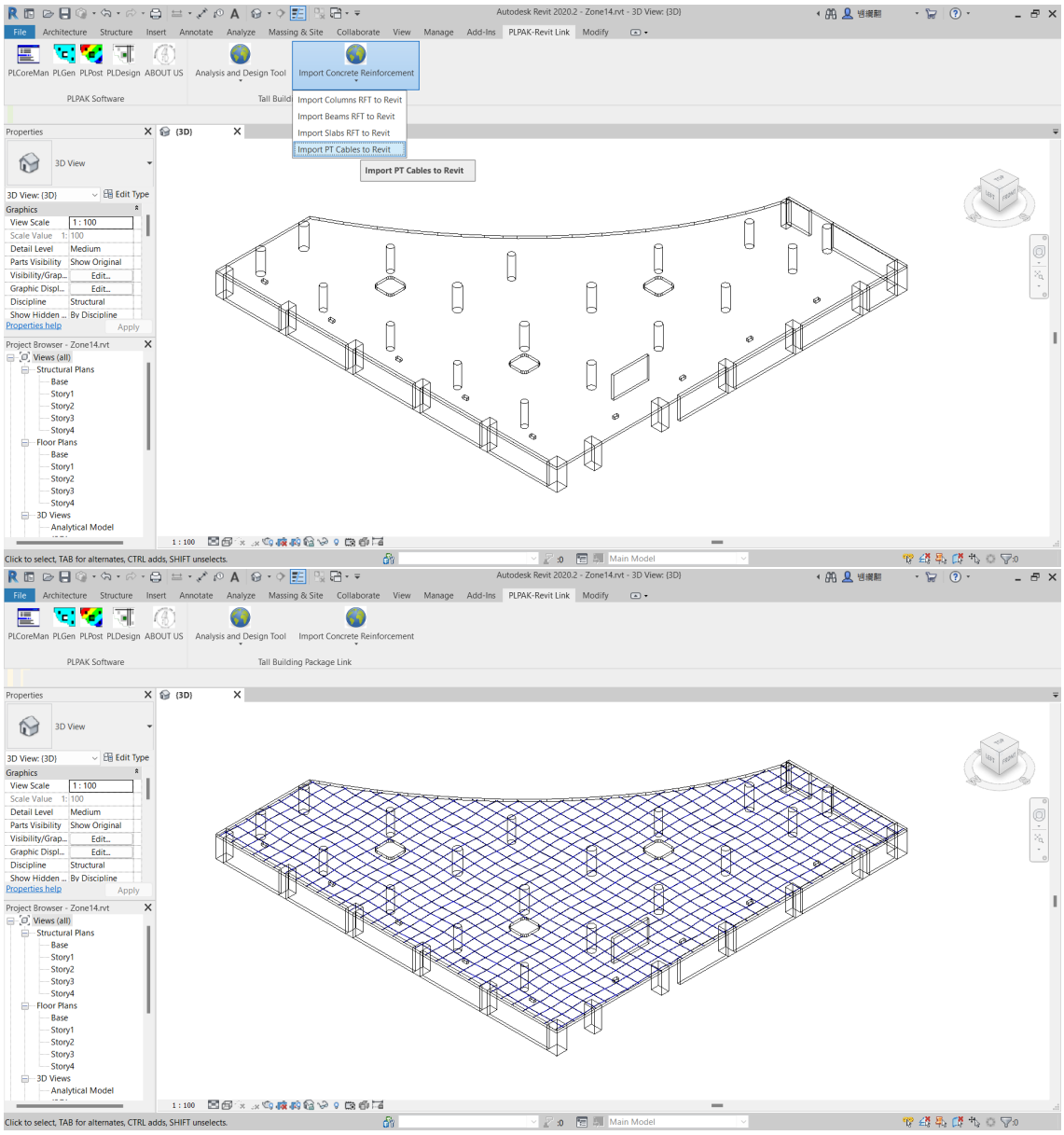
- To export cables the user should click on File, then click on Export design data.



- Enter the name of the level containing the slab in Revit and click Export to save the file.



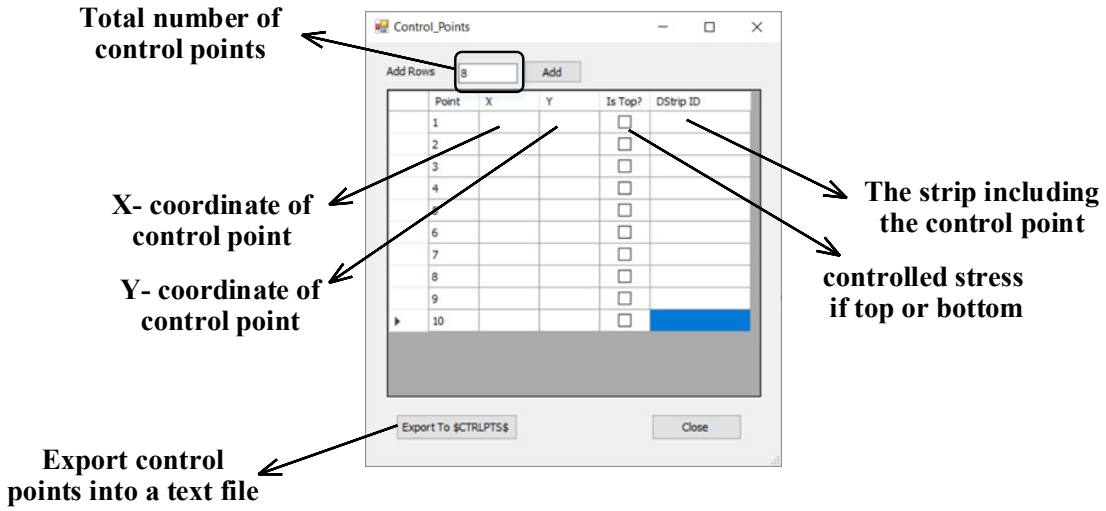
- Open the Revit model and click on Import Concrete Reinforcement, then click on Import PT Cables to Revit and choose the saved file.



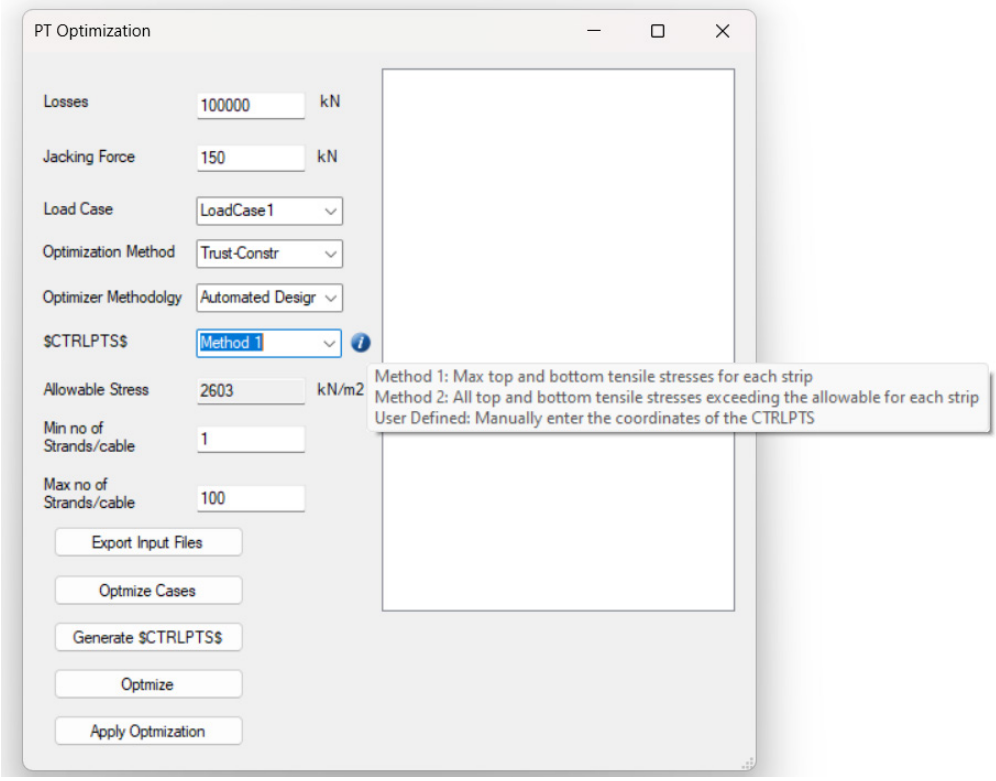
- **Hint: The following families must be loaded in Revit (pt family5.rfa and M_Structural Framing Tag.rfa). These families can be found in the following path (C:\Program Files\PLPAK\PLDesign\PTPAK)**

11.Optimization tool (Post-Tension Optimization Tool is required for this option)

- To use the optimization methodology each PT design strip must contain only one cable in its center. After that the optimum number of strands in this cable can be divided into several cables inside this strip to satisfy the design code requirements.
- This optimizer calculates the minimum number of strands in each cable that satisfies no violation of stresses at certain chosen slab points (Ctrl Points) under a certain load combination. These Ctrl Points should be defined. It is recommended that these points are the points having peak tensile stresses that exceed the allowable stress.
- The user can define these Ctrl Points manually by clicking on Define Ctrl Points button, enter number of the points, there coordinates, and the Strip ID. Note, these CTRL points can be generated automatically as illustrated in the next step.



- The user should open the PT Optimization wizard by clicking on the Optimize Cables button. Hence the user should define average losses, jacking force, the target load case/combination, optimization method, optimizer methodology, CTRL points input methodology, allowable stress, minimum and maximum number of strands per cable. It has to be noted that CTRL points input methodology can be defined using 3 ways, Method 1: Max top and bottom tensile stresses for each strip, Method 2: All top and bottom tensile stresses exceeding the allowable for each strip, User Defined: Manually enter the coordinates of the CTRLPTS using the previously mentioned method.
- Click on Export Input Files.
- Click on Optimize cases and wait till the optimizer ends.
- Click on Optimize and wait.
- Click on Apply optimization.
- Run the model again and check that service stresses are safe and optimum for all design strips.



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