



DEEPFEM - FINITE ELEMENT ANALYSIS IN DEEPEX

USE OF DEEPFEM

DeepEX software program

Document Version 1.0.2.1

Issued: 27-Sep-2019

Last Review: 10-Oct-2019



www.deepexcavation.com

www.deepex.com

Table of Contents

1. Introduction	3
2. Applying an Analysis method to a DeepEX model	4
2.1. Conventional Limit Equilibrium Analysis Method (LEM).	4
2.2. Non-Linear (Beam on elastoplastic foundations) Method (NL).....	5
2.3. Limit Equilibrium and Non-Linear Analysis Combination Method (LEM+NL).....	5
2.4. Finite Element Analysis Method (FEM).....	6
2.5. Limit Equilibrium and Finite Element Combination Method (LEM+FEM).	6
3. Defining the Mesh Analysis Options	7
4. Defining Convergence Options	9
5. Defining Soil Properties for FEM Analysis	10
5.1. General Soil Properties	11
5.2. Soil Models and Behavior Properties.....	12
6. Defining Wall Friction to the Model.....	13
7. Common Reasons why a FEM Analysis might not Converge	14
8. Performing FEM Analysis	17
9. Finite Element Analysis results in DeepEX	17
10. Example 1: Braced Excavation - LEM, Non-Linear and FEM Analysis Comparison.....	21
11. Example 2: Slope Surface close to a Building: Slope Stability and FEM Settlements	26
12. Example 3: Stepped Anchored Walls – FEM Analysis	31

1. Introduction

The Finite Element Method (FEM) is a numerical method for solving engineering problems. Over the last years, it is adopted by several structural and geotechnical design software programs and it offers an advanced solution to complex soil retaining problems. FEM uses mesh generation techniques for dividing a complex problem into smaller parts, called finite elements, that can be described by simpler equations. Usually, a Finite Element software is used in order to assemble the finite elements and simulate the entire model.

DeepEX software uses our Finite Element Analysis method engine, called DeepFEM. This option is implemented in all software versions since 2019, and it can be activated as an additional, optional module within the software, allowing us to model full soil-structure interaction considering all construction stage effects. The following options are available:

- Elastic and inelastic soil constitutive laws with associative or non-associative Failure Criteria (e.g. Mohr-Coulomb, Drucker-Prager, Soil Hardening).
- Drained and Undrained modelling of clays with excess pore pressures.
- External loads: Prescribed, surface Strip and Linear Surcharges, 3D Loads (Buildings, Footings, Area Loads).
- Automated, high-quality mesh generation with quadratic triangular finite elements.
- Inelastic zero thickness interface elements automatically generated on soil to structural element interfaces.
- Steady state seepage analysis where the user can define boundary conditions, drains and wells.
- Expert decision making on the estimation of the FEM analysis parameters.
- Calculation of Surface Settlements.
- Deadman Wall Systems.
- Design of Anchored walls and Braced excavations.
- Analysis of Multiple walls.

With the Finite Element Analysis method activated, DeepEX offers a complete set of all analysis methods in deep excavation. The purpose of this document is to explain how the FEM analysis method can be defined and used in DeepEX.

2. Applying an Analysis method to a DeepEX model

In the Analysis tab of DeepEX, we can select which method we wish to apply to a DeepEX model. We can define different analysis methods in different construction stages in the same software file.

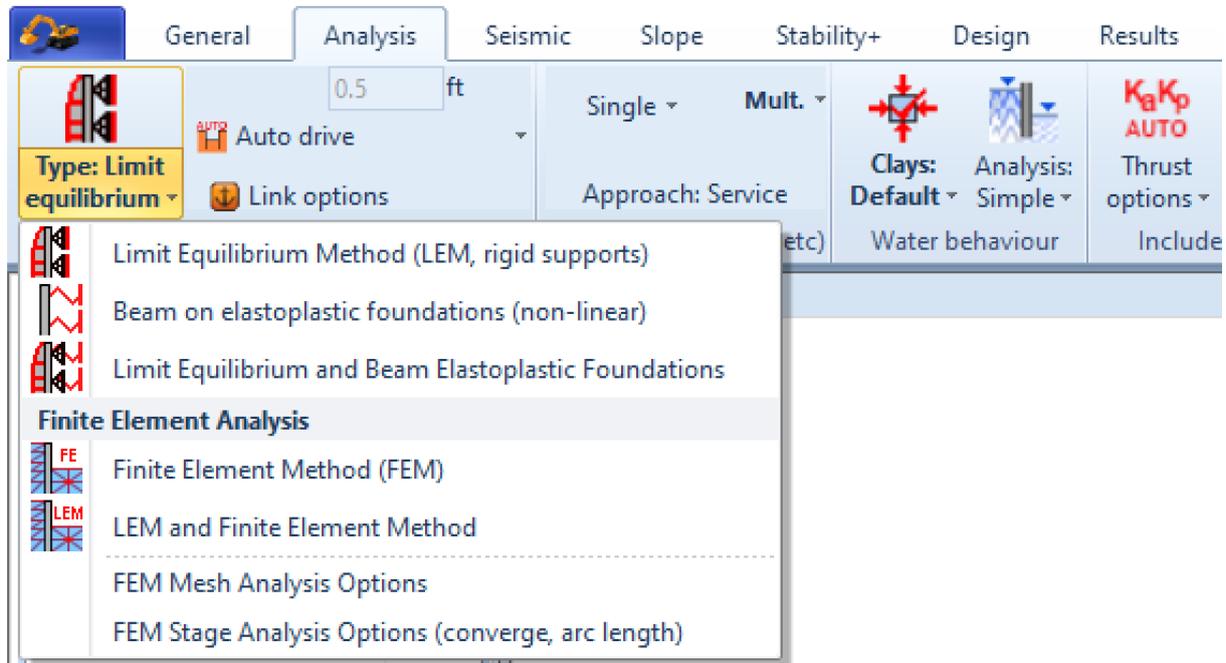


Image: DeepEX – Define Analysis Method

The following sections present the available analysis methods in DeepEX software.

2.1. Conventional Limit Equilibrium Analysis Method (LEM).

Limit equilibrium is an analysis method, where limit state conditions are assumed. For excavations and earth retaining structures this usually means that earth pressures are assumed on both the retained and excavated sides. These pressures may represent a failure state such as active or passive lateral earth pressures, or an assumed redistribution such as diagrams by Peck or FHWA.

In Limit Equilibrium Analysis, the retaining wall is analyzed to provide moment and force equilibrium, when possible. Support reactions are also calculated, typically by using the tributary area method.

2.2. Non-Linear (Beam on elastoplastic foundations) Method (NL).

DeepEX implements a non-linear finite element code for the analysis of the mechanical behavior of flexible earth retaining structures during all the intermediate steps of an open excavation. The non-linear engine is empowered by many unique advanced features. DeepEX offers the following elastoplastic soil models:

- a) Linear elastic - perfectly plastic
- b) Hyperbolic soil model
- c) Subgrade reaction soil model
- d) Small Strain Hardening model

On the reloading part, every soil model has a linear reloading elasticity parameter. Such a parameter should typically range from 2 to 4 times the loading elasticity value (with average 3). In excavations, the reloading elasticity parameter typically describes the remaining soil below the excavation while the loading elasticity is mostly applicable for soil on the retained side. In a non-linear analysis the excavation models reduced to a plane problem, in which a unit wide slice of the wall is analyzed, as outlined in the Figure below. Therefore, DeepEX is not suitable to model excavation geometries in which three-dimensional effects may play an important role. In the modelling of the soil-wall interaction, the very simple yet popular Winkler approach is adopted. The retaining wall is modelled by means of beam elements with transversal bending stiffness EI ; the soil is modelled by means of a double array of independent elastoplastic springs; at each wall grid point, two opposite springs converge at most.

2.3. Limit Equilibrium and Non-Linear Analysis Combination Method (LEM+NL).

In this case, DeepEX will first use the LEM method in order to calculate the wall embedment safety factors, and then will run the analysis with the soil springs in order to calculate all other parameters (support reactions, soil pressures, wall moment and shear stresses, wall displacements etc.).

In stepped excavations and deadman wall systems, a part of the passive pressures of the back wall is transferred to the front wall as an active impact load when the LEM or the LEM+NL methods are used. The magnitude of this additional load is affected by several parameters, like the depth of the back wall (thus the passive pressures), the distance between the two walls etc.

2.4. Finite Element Analysis Method (FEM).

FEM analysis can consider all construction stage effects and enables us to model full soil-structure interaction. Soil is modelled with a mesh of quadratic triangular finite elements. DeepEX does all the stiffness calculations and helps us to estimate FEM analysis parameters. In FEM, the soil model of each soil type can be defined easily. DeepEX has implemented several models like Mohr-Coulomb, Soil Hardening, Cam Clay and more. It considers drained and undrained clay behavior and it can perform water flow analysis.

DeepFEM can be used within the DeepEX software interactive interface, to analyze composite models like braced excavations with struts and rakers. Anchored walls. Deadman wall systems and more. It can calculate all analysis results – soil and water pressures, wall moment shear and displacement diagrams, support reactions, structural and geotechnical ratios, surface settlements and more. The results can be presented in tables and graphically on the model area for each stage.

NOTE: The additional FEM Analysis module is required for this option.

2.5. Limit Equilibrium and Finite Element Combination Method (LEM+FEM).

In this case, DeepEX will first use the LEM method in order to calculate the wall embedment safety factors, and then will run the analysis with the finite elements in order to calculate all other parameters (support reactions, soil pressures, wall moment and shear stresses, wall displacements etc.). In stepped excavations and deadman wall systems, a part of the passive pressures of the back wall is transferred to the front wall as an active impact load when the LEM or the LEM+NL methods are used. The magnitude of this additional load is affected by several parameters, like the depth of the back wall (thus the passive pressures), the distance between the two walls etc.

NOTE: The additional FEM Analysis module is required for this option.

3. Defining the Mesh Analysis Options

In the Analysis tab of the software, we can select to edit the mesh analysis options for the Finite Element Method:

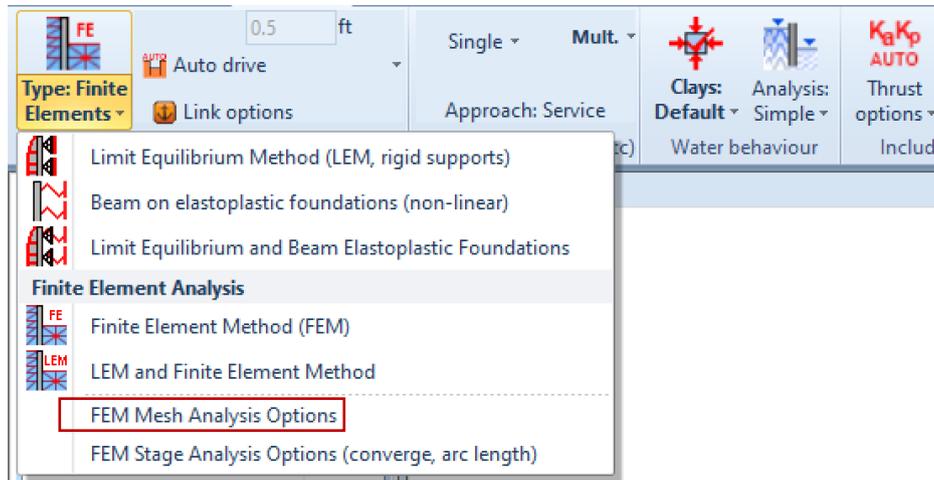


Image: DeepEX – Define Mesh Analysis Options

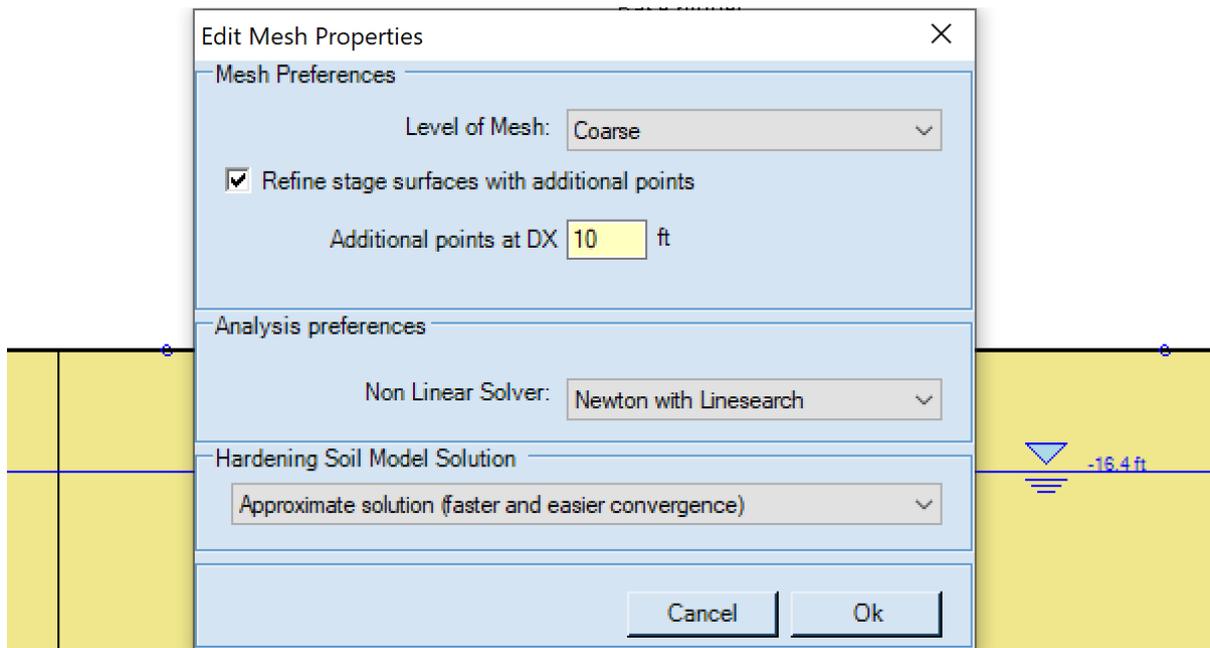


Image: Edit Mesh Properties Dialog

In the dialog that appears (see Image above), the following options are available:

- **Level of Mesh**

This option defines the level of refinement of the generated mesh (size of finite elements on the model area). The level of mesh can be Coarse, Medium or Fine. This option affects the finite element convergence and accuracy of results. A Fine Mesh would produce more accurate results, but the analysis takes longer.

- **Option to Refine stage surfaces with additional points**

With this option selected, further mesh refinement is imposed on the soil layer boundaries and soil body surface of each stage. The length variable DX corresponds to the maximum finite element dimension allowed on the aforementioned locations.

- **Non Linear Solver**

With this option the user can select the preferred algorithm for the solution of the algebraic non-linear equation system formed at each stage (or sub-step of stage) by the fem method. When exponential constitutive laws are used, the option of Newton with line-search is selected by default. The available options are:

- ✓ **Newton:** This option corresponds to the traditional Newton-Raphson method.
- ✓ **Newton with Linesearch:** This option corresponds to the Newton-Raphson method where the solution of each step is appropriately scaled by a line-search approach in accordance to the Wolfe conditions.
- ✓ **Modified Newton:** This option follows a similar approach to the Newton-Raphson method with the only difference of using the elastic stiffness matrix instead of the tangent stiffness matrix for the calculation of the displacement increment.

- **Hardening Soil Model Solution**

This option defines the Hyperbolic Hardening model that will be used in FEM. For more information, see Section 5 below. The following options are available.

- ✓ **Approximate Solution** – This option makes the analysis faster and easier to converge
- ✓ **Exact Theoretical** – This option makes the analysis more accurate but slower and more difficult to converge

4. Defining Convergence Options

In the Analysis tab of the software, we can select to edit the stage analysis options for the Finite Element Method:

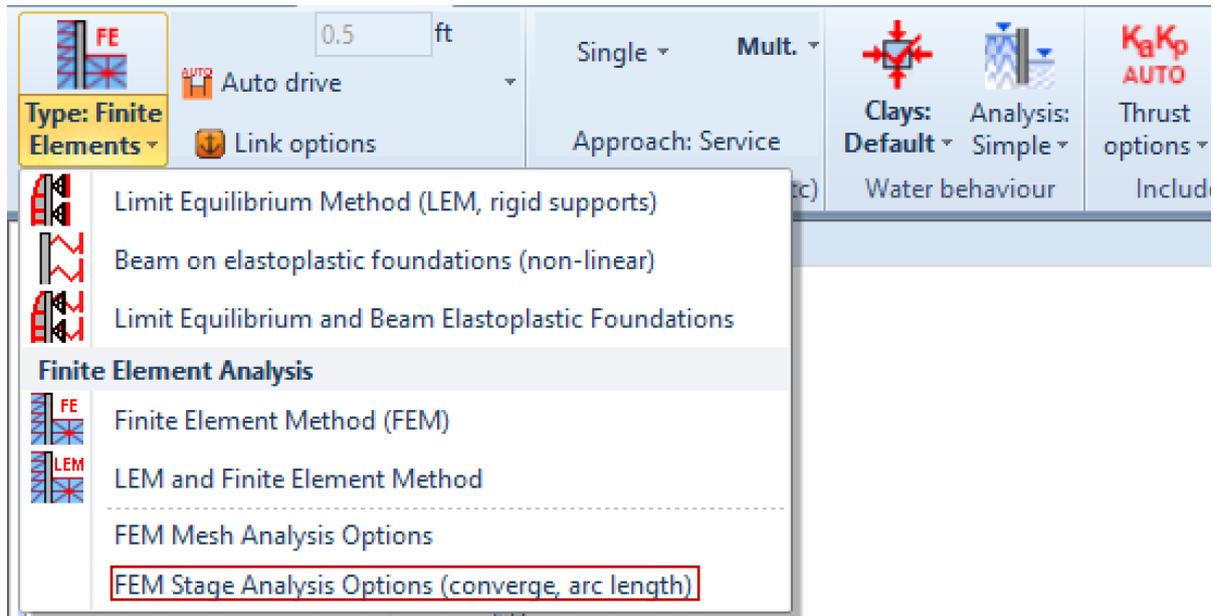


Image: DeepEX – Define Analysis Convergence Options

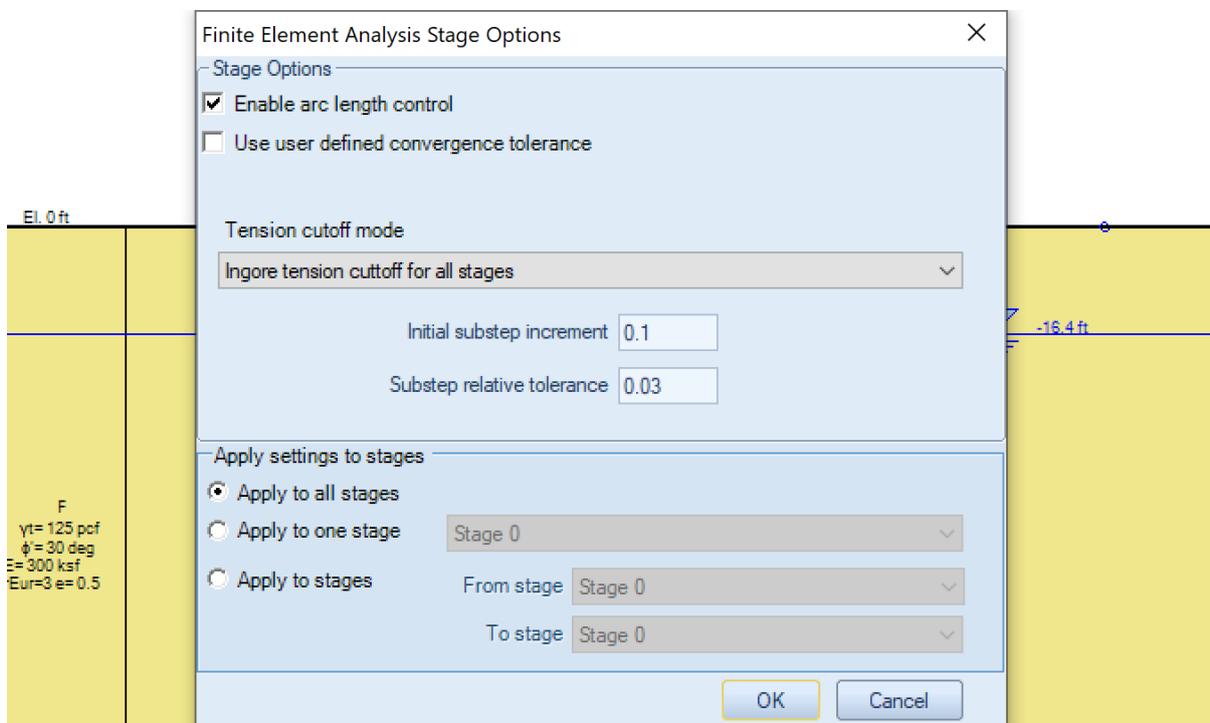


Image: Edit Stage Convergence Options Dialog

In the dialog that appears (see Image above), the following options are available:

- **Option to Enable/Disable arc length control**

This option enables the Crisfield's Formulation of the Arc-length method for the solution of the nonlinear equation system. The use of the arc-length method could provide better convergence in problems with snapping behavior (work softening behavior of the constitutive laws).

- **Option to define a user-defined convergence tolerance**

By selecting this option, we can define a custom convergence to our model analysis. If not selected, the software will use the default tolerance, which is 3%. Any value more than 3% should be used with caution.

- **Option to Use or Ignore tension cutoff in all stages**

This option, when selected, imposes a tension cut-off on all soil constitutive laws. (i.e. principal stresses are always in compression).

- **Define the Initial substep increment and the substep relative tolerance**

This option enables the use of sub-stepping in the solution of each stage. More specifically, the unbalanced forces calculated upon the initiation of each stage, are proportionally eliminated in multiple steps. This option aids on the easier convergence of a stage, as each sub-step converges on the equilibrium path and scenarios of divergence from the path are avoided. The user can select the initial step increment, and when necessary the software will further decrease it. Additionally, the user can select the normalized unbalanced force tolerance for each sub-step. The When exponential constitutive laws are used, this option is selected by default.

5. Defining Soil Properties for FEM Analysis

In the General tab of DeepEX we can access and edit the soil properties. In the dialog that appears, we can create a list of soils we need to use in the specific software file and define the exact soil properties for each soil in the list. The following sections present the soil properties in DeepEX that affect the Finite Element Analysis method.

5.1. General Soil Properties

In the **General tab** of the Edit Soil Types dialog, we can select the soil type (sand, silt, clay (drained or undrained), gravel, intermediate geo-material or rock) and define the general soil properties for each soil (unit weights, cohesion, friction angle etc). If we select and Undrained behavior for the clay, the software will use the defined undrained shear strength S_u and set internally the friction angle to 0. If we select a drained behavior, DeepEX will use the defined cohesion c' and the defined friction angle.

NOTE: For all soil layers (included sands) it is recommended to use a minimum cohesion value of 5 psf (or 0.25 kPa). This is extremely beneficial to the FEM model convergence. If the model is not converging, we suggest to try to increase the sands cohesion value.

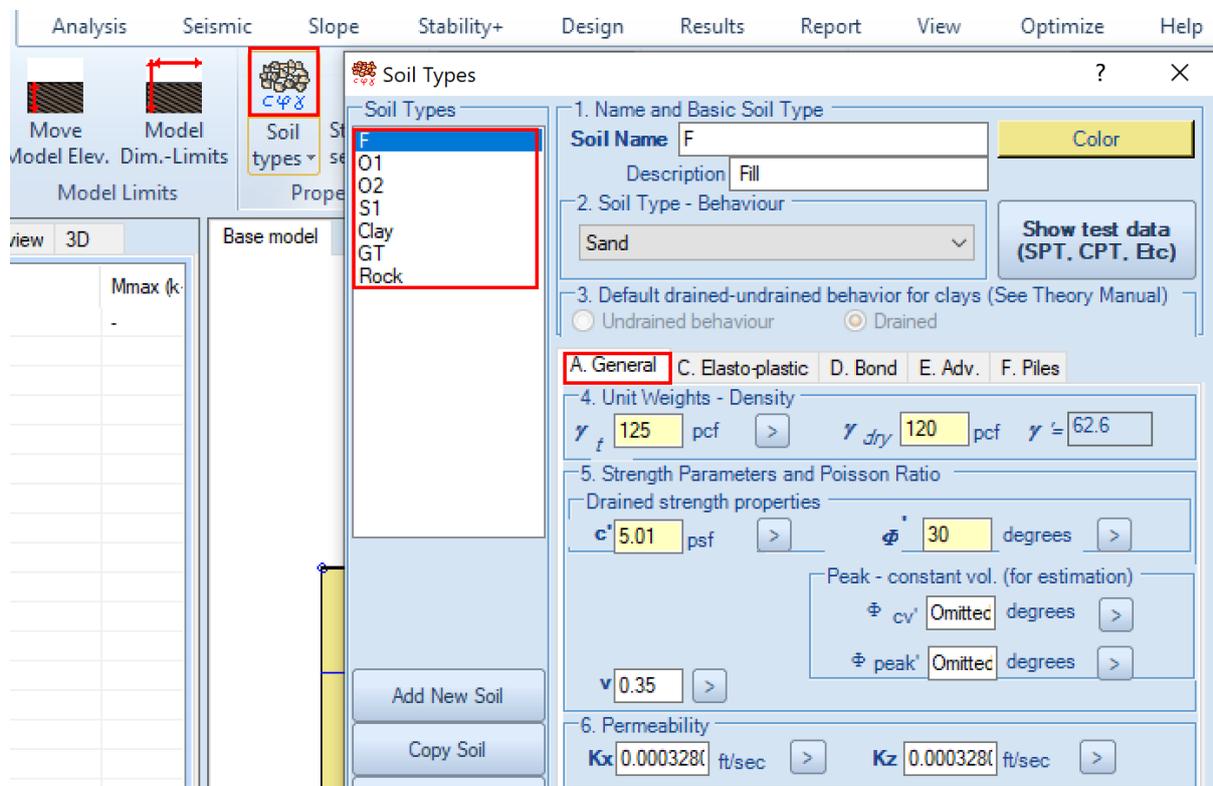


Image: Access Edit Soils dialog and Define General Soil Properties

5.2. Soil Models and Behavior Properties

In the **Elasto-plastic tab** of the Edit Soil Types dialog, we can select the soil model (used in the Non-Linear analysis) and define the soil model properties for each created soil. When the Finite Element Analysis method is selected in a software file, the selected Soil Model is converted to an equivalent FEM model as follows:

Elastoplastic Model: Single stiffness model with linear elasticity in combination with a non-associative Mohr-Coulomb plasticity rule.

- ✓ linear representation of the pre-yielding region.
- ✓ constant E_{50} along the depth.
- ✓ Unloading/reloading young modulus identical to the loading stiffness modulus.
- ✓ constant dilatancy angle ψ .

Exponential (Hyperbolic) Model (approximate solution): Double stiffness model multi-surface model (Mohr-Coulomb yielding criterion).

- ✓ Two-piece linear representation of the pre-yielding region.
- ✓ Exponential E_{50} relative to the minor principal stress σ_3 . (stress dependent stiffness modulus).
- ✓ Individual formulation of Unloading/reloading young modulus E_{ur} .
- ✓ constant dilatancy angle ψ .

Exponential (Hyperbolic) Model (complete solution): Soil hardening model

- ✓ hyperbolic representation of the pre-yielding region.
- ✓ exponential E_{50} relative to the minor principal stress σ_3 . (stress dependent stiffness modulus)
- ✓ Individual formulation of Unloading/reloading young modulus E_{ur} .
- ✓ stress dependent dilatancy angle ψ .

Small Strain Hardening: This model is currently under development for DeepFEM. When this option is selected to any soil, DeepEX uses the Exponential model for the FEM analysis.

Subgrade Modulus: This option does not exist in FEM analysis (it works in the Non-Linear method with the soil springs). When this model is selected, DeepEX uses the Elastoplastic Model in FEM analysis.

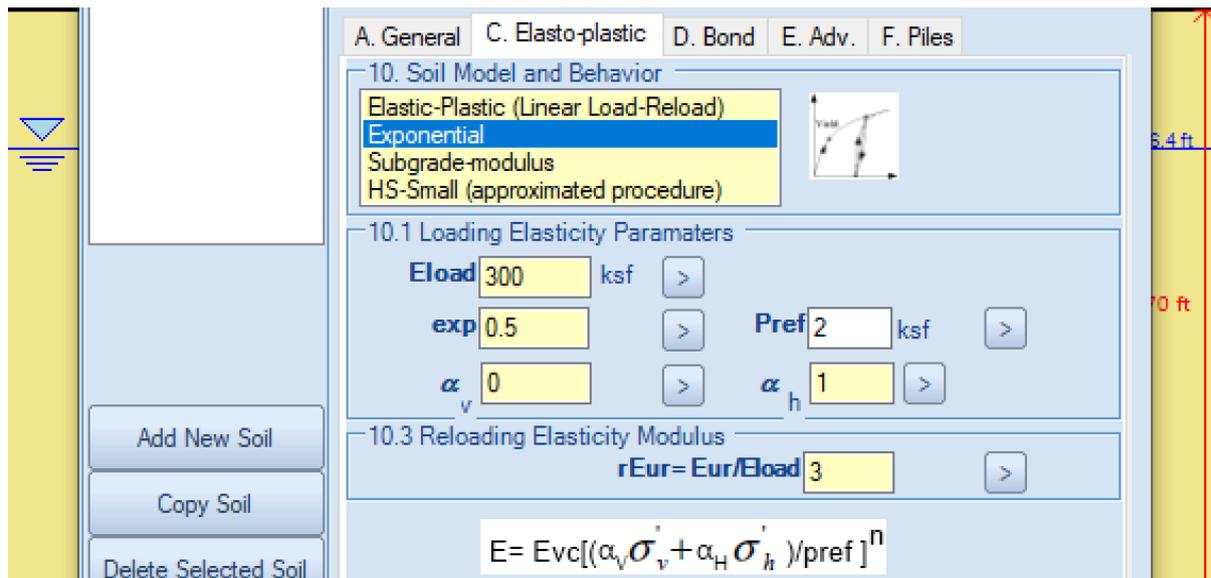


Image: Define Soil Model in DeepEX

6. Defining Wall Friction to the Model

When the Finite Element Analysis is selected in a DeepEX model, we usually have to define some wall friction to the model. Typically, the wall friction should be defined as a percentage of the soil friction for each wall. This option is implemented in the Analysis tab of DeepEX. In the same area, we can find the option to include a different percentage of soil friction and a wall friction on each wall side (active and passive).

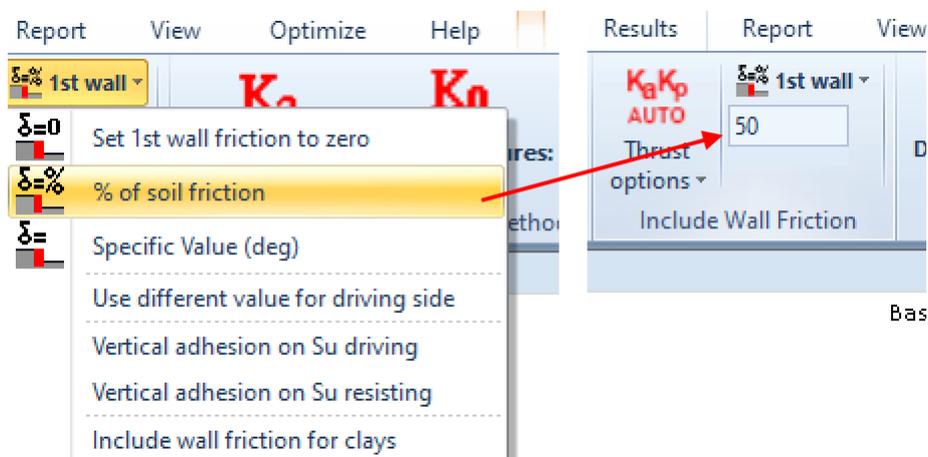


Image: Define Wall Friction in DeepEX

NOTE: A use of a high wall friction percentage is beneficial to the FEM convergence. Typically, we should use 30 to 70% of the soil friction.

7. Common Reasons why a FEM Analysis might not Converge

Sometimes, a model analysis in DeepEX with the use of the Finite Element Analysis method might fail. In this case, the software returns the message “Not Analyzed” for the respective design section. Most commonly, FEM fails because of project geometry, staging, user assumptions or soil properties reasons. Below we present some common FEM failure reasons and the actions we should take in order to avoid them and make the model converge.

- **Project Geometry**

In DeepEX, we should avoid totally vertical soil cuts. When defining the soil surface points, we have to make sure that we use slopes. In general, an unsupported slope surface angle cannot exceed the soil friction angle. If the cut is totally vertical, or quite bigger than the soil friction angle, the model might collapse before the analysis goes through.

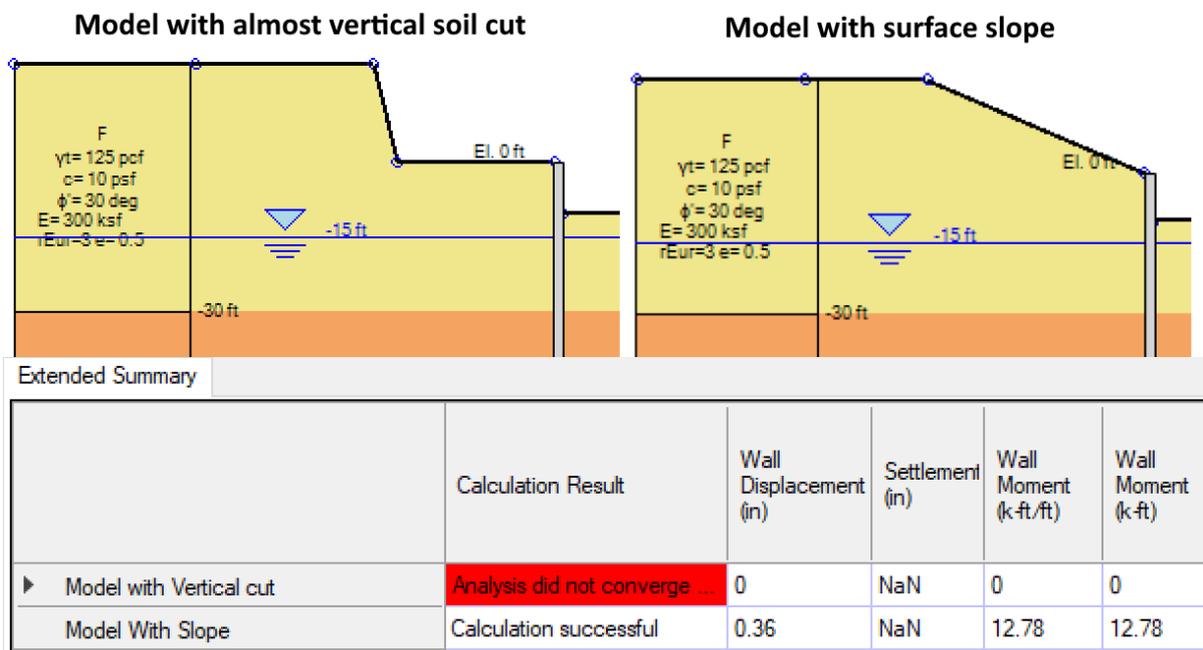


Image: Model with almost vertical cut VS Model with Slope

- **Construction Stages**

In DeepEX, we recommend the use of multiple construction stages for all analysis methods. This allows us to review the results in all construction stages and take important decisions about how to effectively optimize our models (increase/decrease structural sections, move support locations etc.).

Apart from model optimization reasons, staging is important for the convergence of our advanced analysis methods (Non-Linear and Finite Element Analysis). In these methods, an initial stage with no excavation on either wall side should be used (the soil surface points on the 2 sides of each wall should have the exact same elevation coordinate Z). After the initial stage, we recommend the use of strict staging, where only a basic action should be executed (excavation, support installation, backfill operation etc.). Dewatering and external load activation can be performed in any construction stage, along with the basic action.

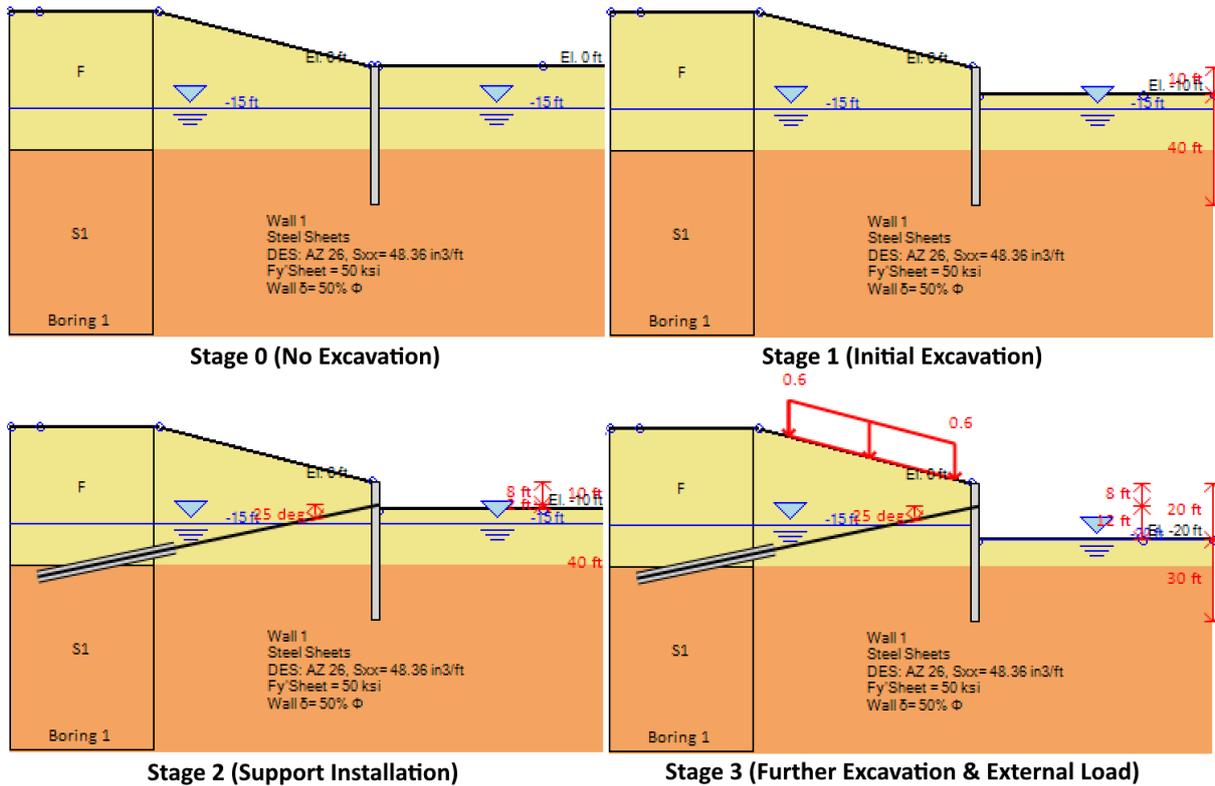


Image: Construction Stages in DeepEX

- **Soil Properties (Cohesion)**

As mentioned in Section 5.1 above, all soils in the model area should have at least some cohesion, when the Finite Element Analysis method is used. We would recommend a minimum of 5 psf (or 0.25 kPa) for all sand soils. If the model does not converge in FEM, we can try to increase slightly the defined cohesion value for the sand soils.

- **Wall Friction**

As mentioned in Section 6 above, the use of high values of wall friction (30 to 70% of the soil friction) are recommended (and in most cases required) for the convergence of the FEM analysis.

- **User Assumptions (Convergence and FEM method options)**

In DeepEX, we offer several method and convergence options for the Finite Element Analysis). If a model does not converge, we can try to unselect the option for soil surfaces refinement with additional points and select the soil hardening approximate model (see Section 3 above). In the FEM convergence dialog (Section 4 above), we can select and define a user-defined convergence tolerance.

- **Make sure that the model is well embedded in LEM analysis**

In general, Non-Linear and Finite Element Analysis methods do not check the wall embedment safety factors. For this reason, we have implemented the combination methods (LEM+NL and LEM+FEM).

DEEPFEM – FINITE ELEMENT ANALYSIS IN DEEPEX

If a model is not well embedded according to the Limit Equilibrium Analysis, it is possible to not converge in Non-Linear and Finite Element Analysis. For this reason, it is recommended to analyze the model with Limit Equilibrium Method, optimize it (soil properties, wall depth etc.), and then assign and run a more advanced method (NL or FEM).

While performing LEM analysis, we have to check all construction stages and make sure that the moment and shear diagrams close below the excavation. We also need to check the basal stability and the wall embedment safety factors. Having adequate wall embedment FS is beneficial to the convergence of the NL and FEM analysis.

Wall Toe Safety:	Wall
Min FS=	0.93
FS Embed=	0.93
FS Rot.=	0.93
Req. toe FS=1:	Fail
Toe El. FS=1:	Fail
Note:	Toe has failed
Basal standard	2.213
FS HYD=	2.144

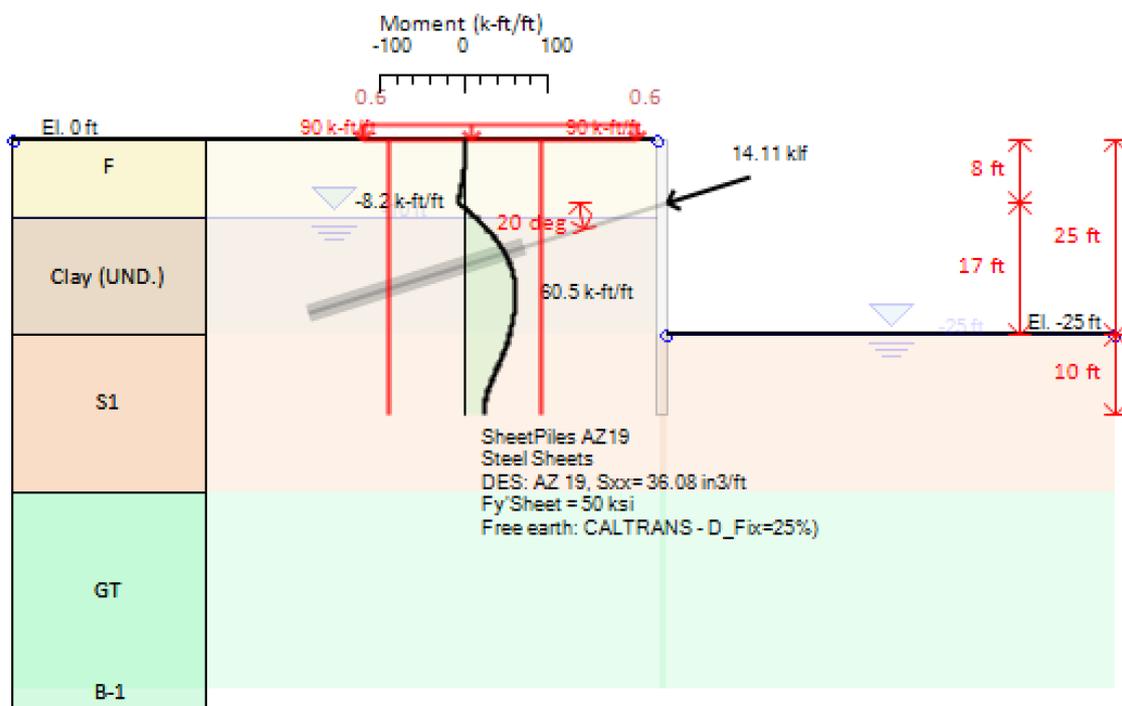


Image: Model – LEM Analysis – Wall Embedment Failure

8. Performing FEM Analysis

During the analysis, all model data are transferred to our FEM analysis engine, DeepFEM, where all calculations take place. User can be informed for the calculation status any time, from the FEM analysis window that appears.

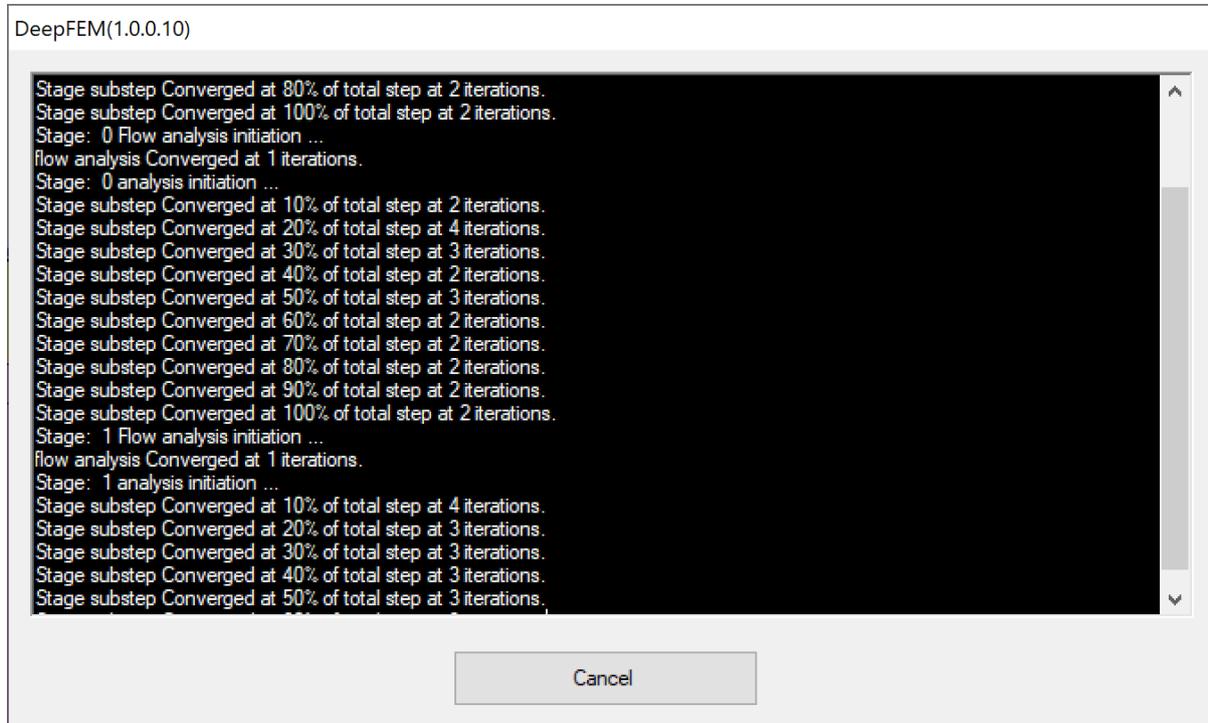


Image: The DeepFEM Analysis Status window

Depending on the model complicity, number of construction stages, number of designed walls and installed support levels, the analysis might take a while longer than the traditional Limit Equilibrium analysis.

9. Finite Element Analysis results in DeepEX

DeepEX can analyze and present results for all construction stages and created design sections, allowing the designer to review the full project history and take important decisions for optimizing a solution while ensuring adequate safety factors. After the analysis is performed, the Analysis and Checking summary table appears, presenting the most critical results for each design section/each construction stage.

On first review, we can see if the analysis converges, if the model is fine (the structural and geotechnical ratios must be below 1), and if there are any notifications (values above the limits are presented in red). The check ratios should be less than 1, with values closer to 1 meaning that an efficient solution is achieved.

Based on these results, we can take some decisions regarding the model manual optimization.

DEEPFEM – FINITE ELEMENT ANALYSIS IN DEEPEX

Extended vs Stage		Calculation Result	Wall Displacement (in)	Settlement (in)	Wall Moment (k-ft/ft)	Wall Moment (k-ft)	Wall Shear (k/ft)	Wall Shear (k)	STR Combine Wall Ratio	STR Moment Wall Ratio	STR Shear Wall Ratio	Max Support Reaction (k/ft)	Max Support Reaction (k)	Critical Support Check	STR Support Ratio	Support Geotech Capacity Ratio (pull out)
▶	Stage 0	Calculated	0	N/A	0	0	0	0	0	0	0	0	0	0	N/A	N/A
	Stage 1	Calculated	0.54	0.24	13.18	13.19	2.12	2.12	0.171	0.1705...	0.049	0	0	0	N/A	N/A
	Stage 2	Calculated	0.41	0.15	8.02	8.02	2.79	2.79	0.104	0.1037...	0.064	4.75	38	0.471	0.352	0.471
	Stage 3	Calculated	0.45	0.33	6.62	6.63	2.94	2.94	0.086	0.0857...	0.068	5.42	43.38	0.537	0.402	0.537

Image: Analysis and Checking Summary – Critical Results/Stage

Several results (wall moment-shear-displacement diagrams, soil and water pressures, net pressure diagrams, support reactions and ratios, surface settlements and more) can be presented graphically, on the model area, for each construction stage. The following images illustrate the calculated graphs, on an anchored wall model where the FEM analysis was performed.

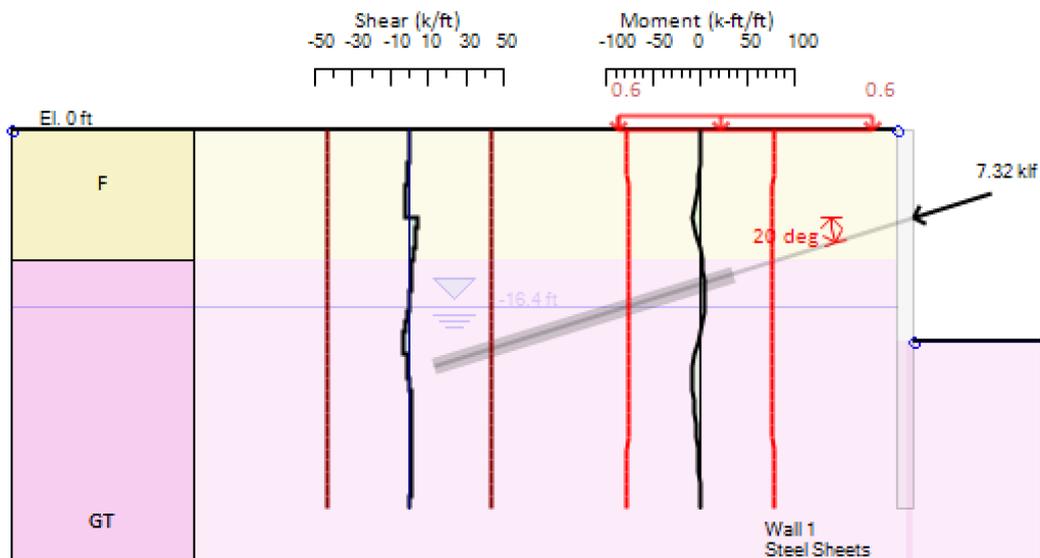


Image: Wall Moment and Shear Diagrams – Support Reaction (FEM Analysis)

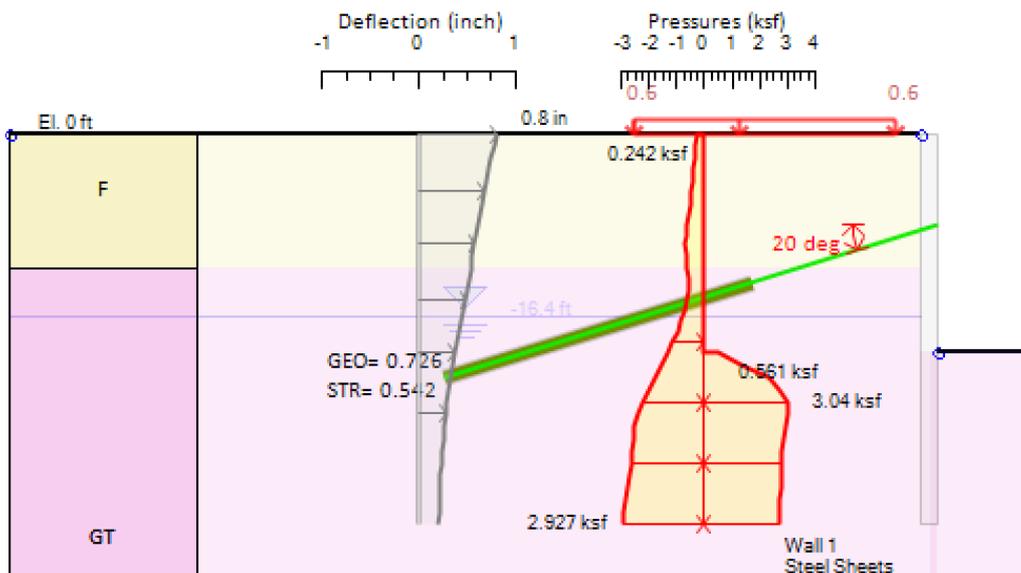


Image: Wall Displacements – Soil Pressures – Support Ratios (FEM Analysis)

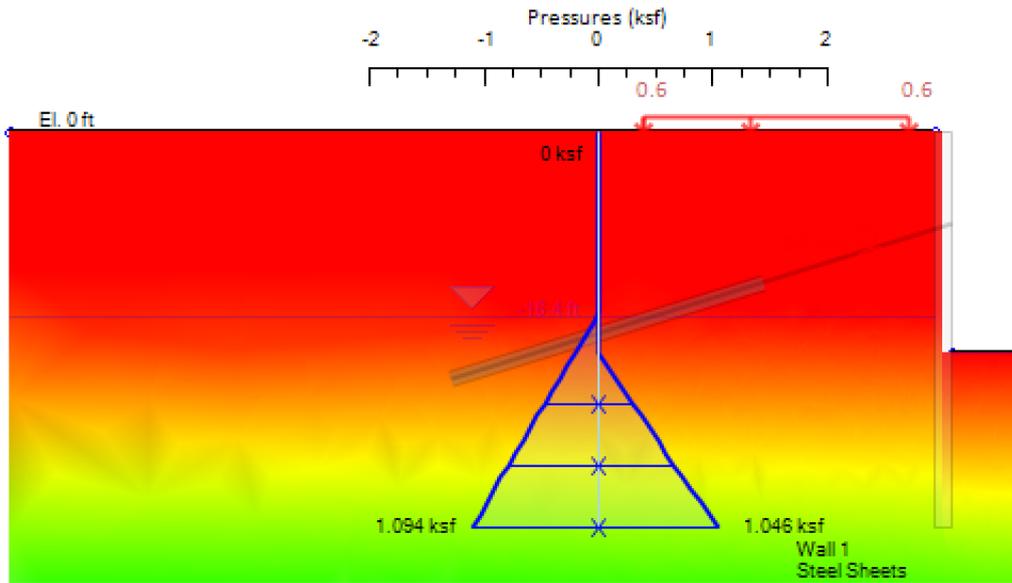


Image: Water Pressures and Water Pressure Shadings (FEM Analysis)

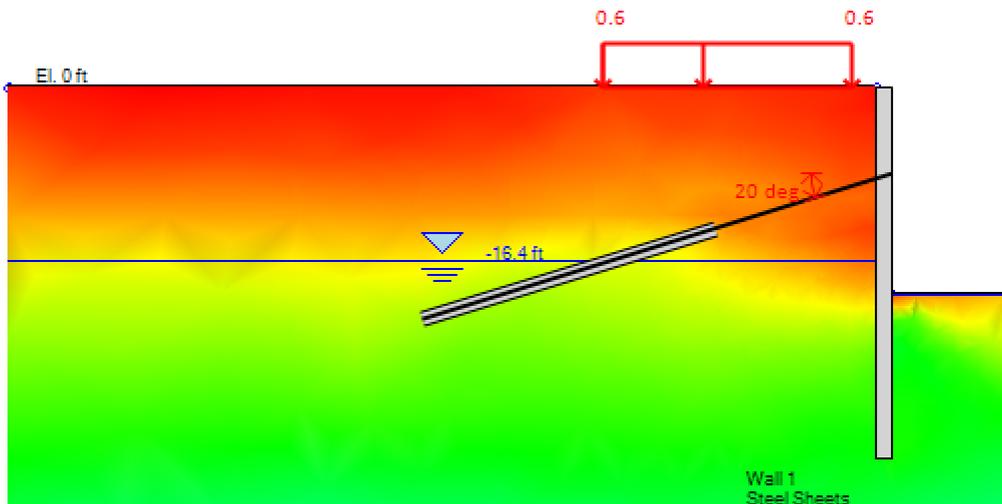


Image: Horizontal Soil Pressure Shadings (FEM Analysis)

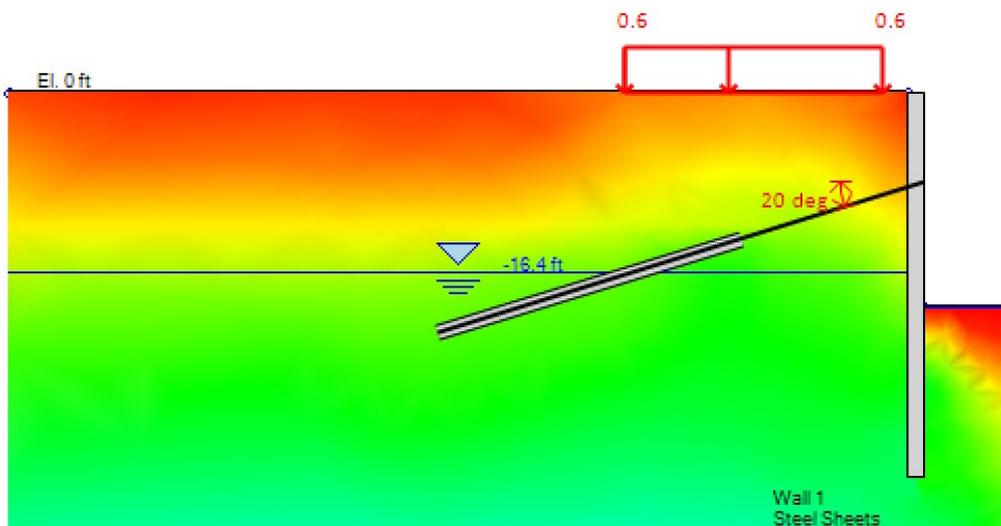


Image: Effective Vertical Soil Pressure Shadings (FEM Analysis)

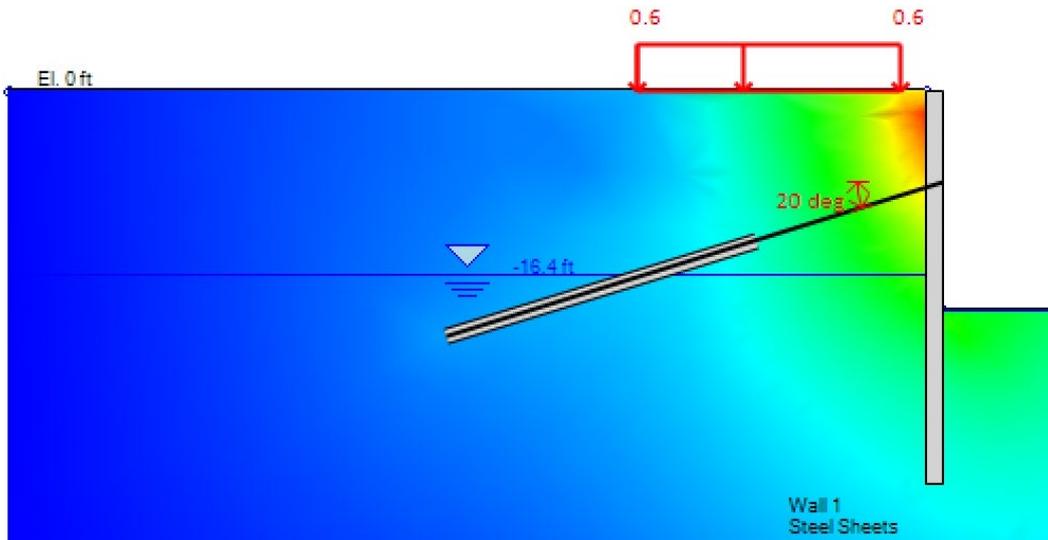


Image: Soil – Structure Horizontal Displacement Shadings (FEM Analysis)

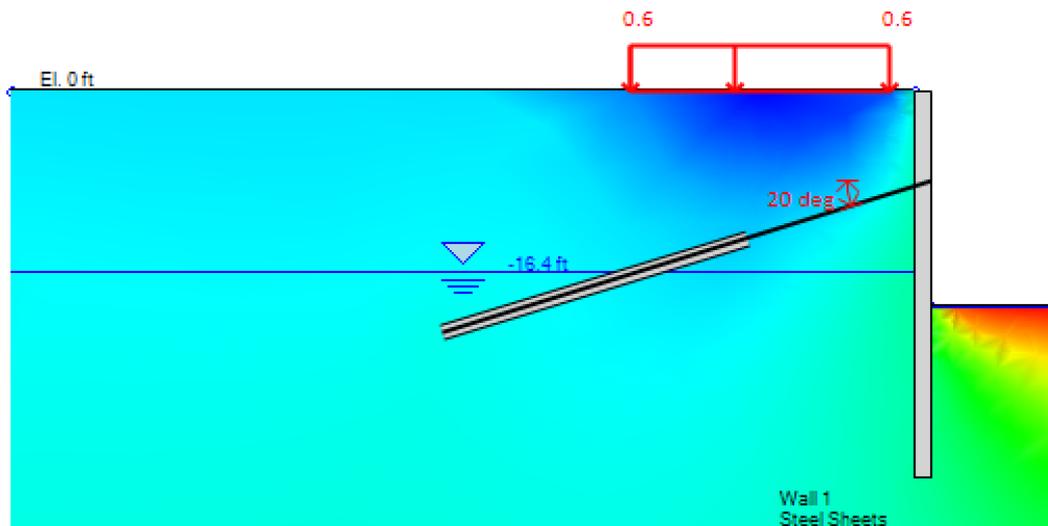


Image: Surface Settlements (FEM Analysis)

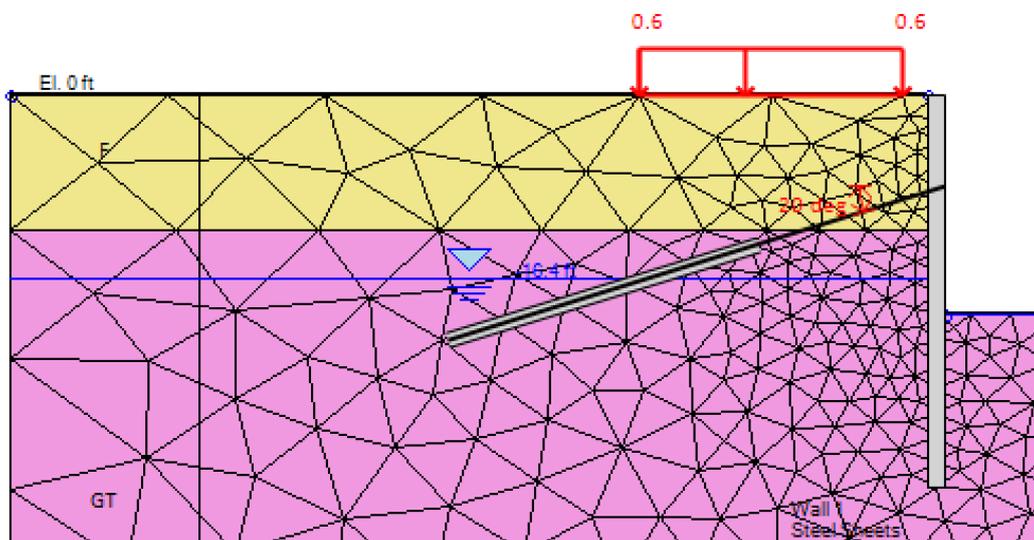


Image: Finite Elements Mesh

10. Example 1: Braced Excavation - LEM, Non-Linear and FEM Analysis Comparison

A. Introduction – Model Description – Soil Properties – Structural Sections

In this example, we will examine a case of a braced excavation system (sheet piles braced with struts), supporting a 30 ft excavation. A 0.6 ksf external load (Strip surcharge, developed for 30 ft, starting 2 ft behind the wall) will be used on the top, on one side of the excavation. The water table is at El: -15 ft.

The model will be generated and analyzed with DeepEX – Shoring Design Software, using all available analysis methods (Limit Equilibrium, Non-Linear and Finite Element Analysis).

The following tables present the soil properties, the wall section properties and the support properties respectively.

Table: Soil Properties and Stratigraphy

Layer Top Elevation (ft)	Soil Type	Dry Unit Weight (pcf)	Total Unit Weight (pcf)	Friction Angle (deg)	Cohesion c' (psf)	Soil Model	Modulus of Elasticity (ksf)	Exponent
0	Sand	125	125	30	10	Exponential	300	0.5
-10	Silt	135	135	34	50	Elastoplastic	550	-

Table: Wall Properties

Top of the Wall Elevation	0
Wall Depth	50 ft
Wall Type	Steel Sheet Piles
Wall Section	AZ 19
Steel Material	A50

Table: Support Properties

Support Level	Elevation	Support Type	Support Section	Steel Material
Level 1	-10 ft	Strut (Pipe)	PP18X0.5	A50
Level 2	-20 ft	Strut (Pipe)	PP18X0.5	A50

B. Modeling in DeepEX: Construction Stages – Analysis Assumptions – Design Standards

The model will be created with the DeepEX model wizard, where we can apply all the project parameters (project type, wall and support structural sections, soil properties and stratigraphy, support locations and design codes). The Wizard generates all construction stages automatically. Then, we can create new Design Sections (copied from the created Base Model), in which we can assign different analysis assumptions, analysis methods and more. In this example we will create a total of 3 Design Sections. Each DS will be analyzed with a different analysis method. This way we can review all and make a method comparison.

Generated Construction Stages

The following image illustrates the model in each construction stage, as they are created from the DeepEX model wizard:

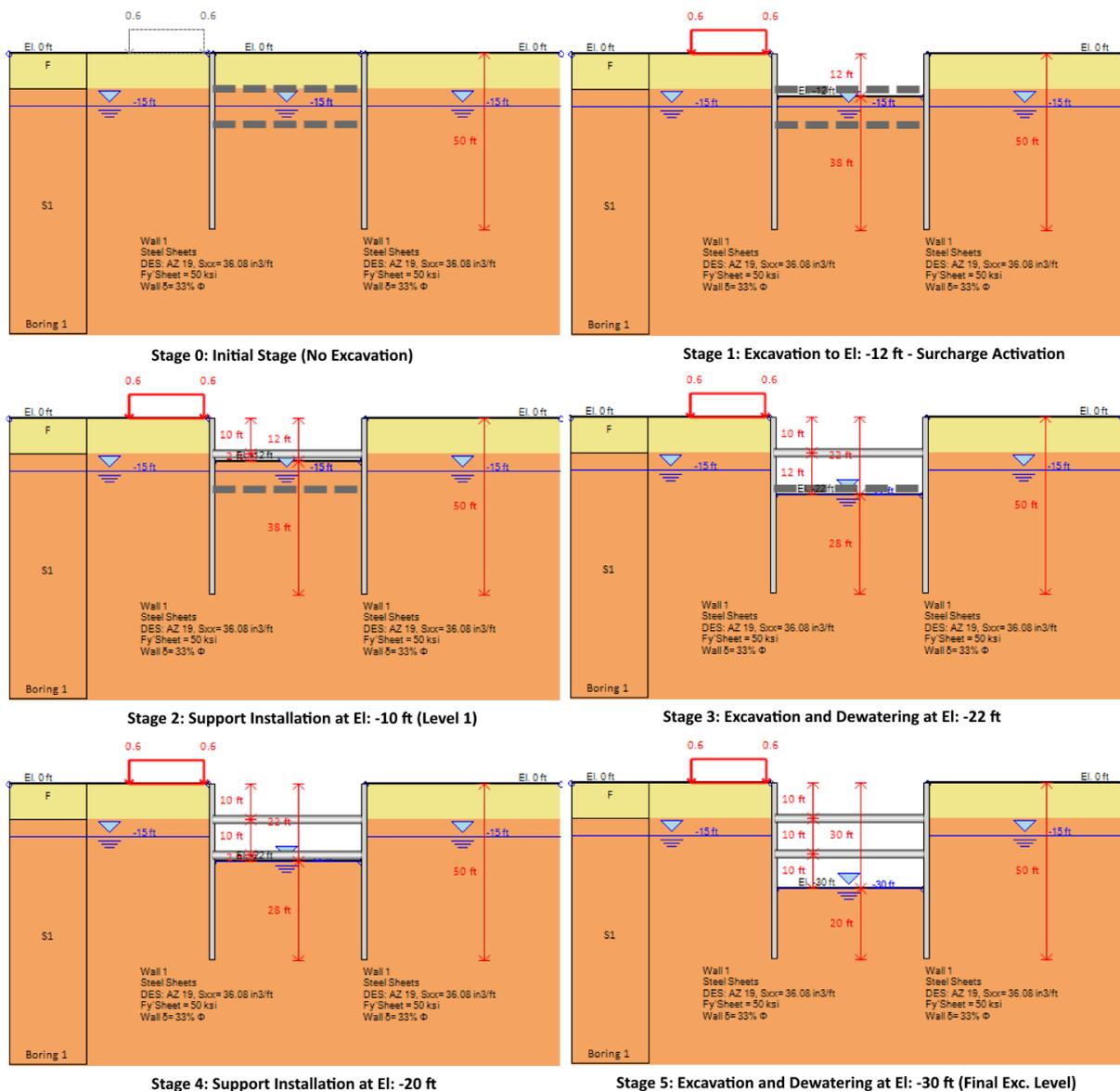


Image: Model in each Construction Stage

Wall Friction and Water Pressures

In this model we will use wall friction on both walls. In the Analysis tab of the software, we select to use on each wall 33% of the soil friction as wall friction.

In this example we will calculate the water pressures using the Simplified Flow analysis method.



Image: Water Pressures method and Defined Wall Friction for each wall

Design Standards

In this model we have used steel sheets and steel struts. In the Design tab of the software, we will select to use AISC 360-16 Allowable method for the calculation of the structural capacities of all steel members.

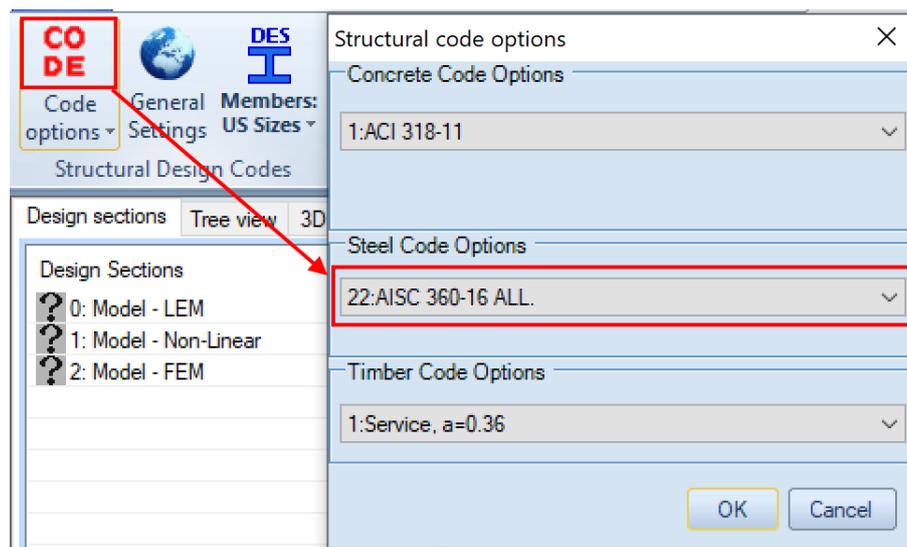


Image: Structural Design Codes

In this example we will perform a Service design, so we will not apply any Geotechnical Standard design combination.

Design Sections – Analysis Methods

We will create new Design Sections as exact copies of the created Base Model. In each design section, we will select to use one of the main DeepEX software methods (Limit Equilibrium, Non-Linear and Finite Element Analysis), and we will define the analysis method assumptions.

- **Design Section 1: Limit Equilibrium Analysis Method**

In this design section, we will select to use the Limit Equilibrium analysis method from the Analysis tab of DeepEX. In the same tab, we will select to use FHWA Apparent pressures for all construction stages with at least one support level activated (in the Cantilever excavation (Stages 0 and 1) we will use Active and Passive pressures). Finally, we will use CALTRANS method for the beam analysis (Wall moments – Displacements).

- **Design Section 2: Non-Linear Analysis Method**

In this design section we will select to use the Non-Linear analysis method (elastoplastic springs) from the Analysis tab of DeepEX.

- **Design Section 3: Finite Element Analysis Method**

In this design section we will select to use the Finite Element analysis method from the Analysis tab of DeepEX. In the Mesh Options dialog, we will select to use the exact theoretical solution for the soil hardening model. In this example we will not use a user-defined convergence tolerance. This value should be edited only when the model does not converge.

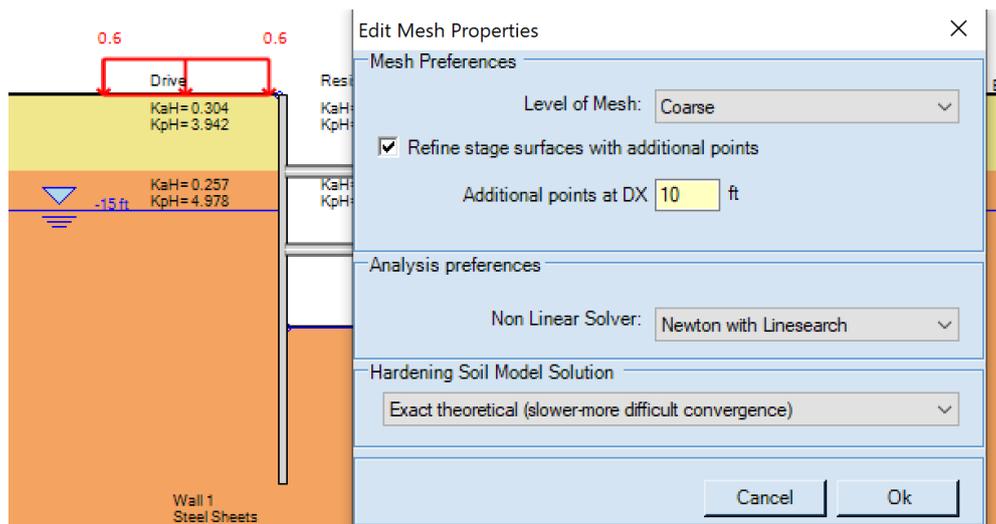


Image: The Mesh Properties dialog – Selected Options

C. LEM, Non-Linear, FEM Analysis Results

DeepEX can perform full structural and geotechnical design of all project items (walls and supports). The calculated results (soil and water pressures, wall moment, shear and displacement diagrams, support reactions, structural and geotechnical check ratios and more) can be presented both in tables and graphically on the model area. DeepEX can analyze and present results for all construction stages, allowing the designer to review the full project history and take important decisions for the most efficient model optimization.

The following images present the analysis and checking summary table (most critical results for each analyzed design section), as well as, some typical result graphs for each analysis method.

DEEPFEM – FINITE ELEMENT ANALYSIS IN DEEPEX

Extended Summary		Calculation Result	Wall Displacement (in)	Settlement (in)	Wall Moment (k-ft/ft)	Wall Moment (k-ft)	Wall Shear (k/ft)	Wall Shear (k)	STR Combine Wall Ratio	STR Moment Wall Ratio	STR Shear Wall Ratio	Max Support Reaction (k/ft)	Max Support Reaction (k)	Critical Support Check	STR Support Ratio
▶	Model - LEM	Calculatio...	0.65	1.86	38.6	38.6	17.35	17.35	0.429	0.429	0.322	14.84	296.77	1.07	1.07
	Model - Non-Linear	Calculatio...	0.79	0.75	27.48	27.48	8.98	8.98	0.305	0.305	0.167	13.44	268.74	0.978	0.978
	Model - FEM	Calculatio...	1.22	NaN	24.34	24.34	8.91	8.91	0.284	0.27	0.165	12.91	258.28	0.944	0.944

Image: Analysis and Checking Summary Table – Critical Results for each Design Section

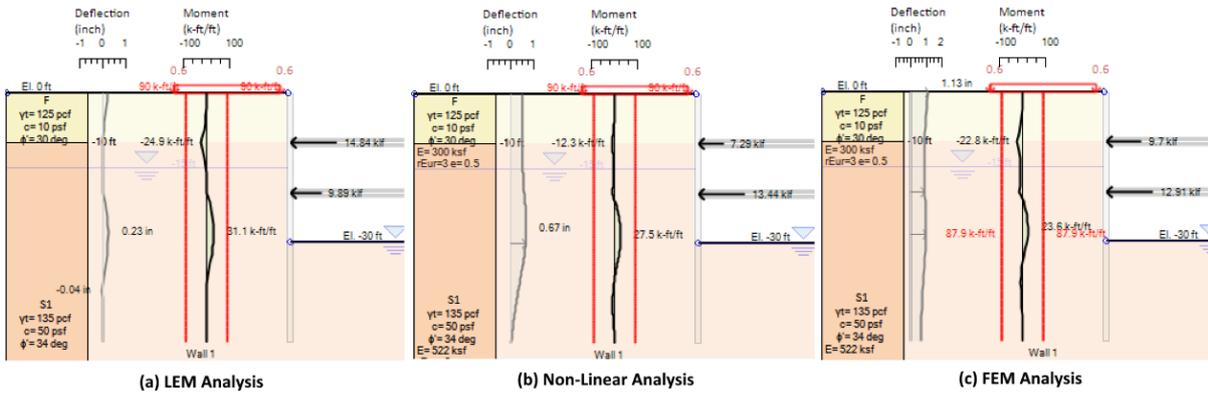


Image: Wall Moments – Wal Displacements – Support Reactions (Stage 5)

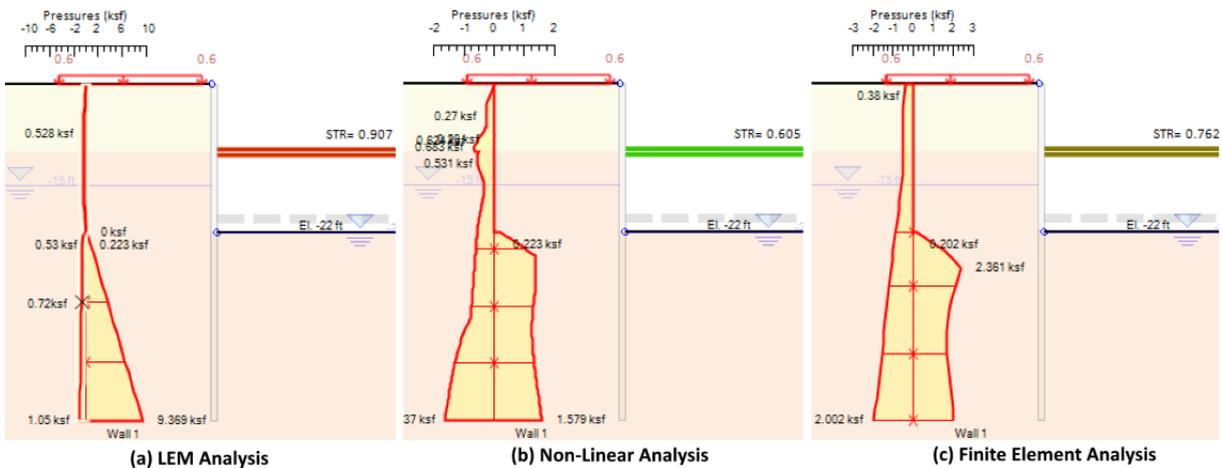


Image: Horizontal Soil Pressures and Support Structural Ratio (Stage 3)

11. Example 2: Slope Surface close to a Building: Slope Stability and FEM Settlements

A. Introduction – Model Description – Soil Properties – Structural Sections

Many times, in urban areas, we have to excavate a steep slope close to existing buildings. In this case, it is recommended to examine both the slope stability, and the surface settlements that might appear below the building. This design example presents a case with a 45deg slope surface. An existing building is located on the top of the bench.

Two soil layers are considered in this model. This example focuses on the slope stability analysis of the slope surface. The following table presents the soil properties and the model stratigraphy.

Table: Soil Properties and Stratigraphy

Layer Top Elevation (ft)	Soil Type	Dry Unit Weight (pcf)	Total Unit Weight (pcf)	Friction Angle (deg)	Cohesion c' (psf)	Soil Model	Modulus of Elasticity (ksf)	Exponent
20	Sand	125	125	30	10	Exponential	300	0.5
10	Till	127	127	36	200	Exponential	600	0.4

For simplicity in this example, a circular slope stability analysis with the Morgenstern-Price (General Limit Equilibrium) method is presented. More complex failure surfaces, possibly using an automatic search approach, should also be examined in real projects.

In the same model, we will perform Finite Element Analysis, so that we can examine the soil stresses, displacements and settlements below the building.

B. Modeling in DeepEX: Building – Analysis Assumptions (FEM and Slope Stability)

In this example, we will not examine any retaining wall, so we can access the wall on the model area and make it considerably shorter, in order not to affect the Finite Element analysis. In this example, we changed the wall depth to 7 ft.

Edit Bench Surface

On the model area of DeepEX, we can right click on the left side of the wall and select to add a left bench surface. In the dialog that appears, we can define the bench slope angle (45 deg), the surface elevation on the bench top (20 ft), and where the bench distance from the wall (1 ft).

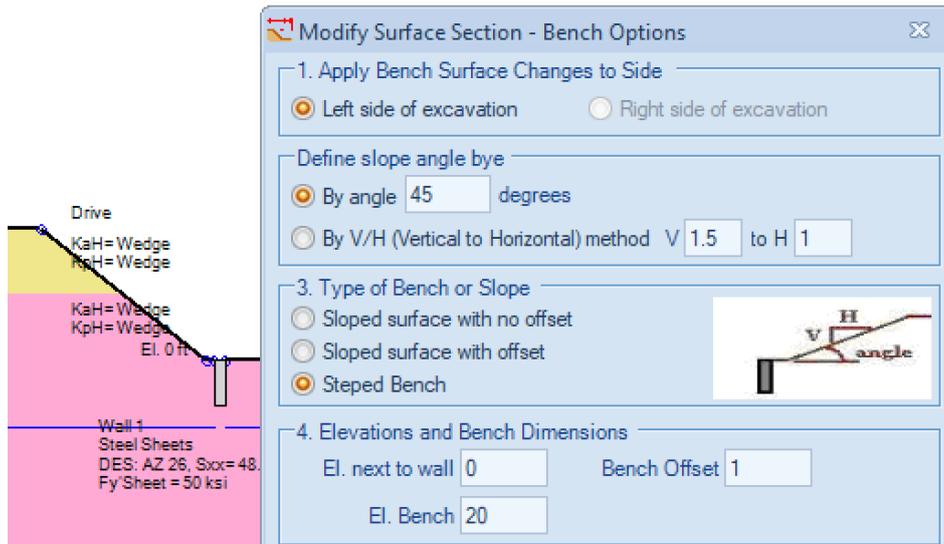


Image: Bench Surface Options

Add a Building

A building can be added as an external load. In the Building Wizard dialog, we can specify the building location, geometry (length, height, number of floors, number of columns per axis), as well as, the dimensions, thicknesses and materials of all building floors, columns, walls and footings.

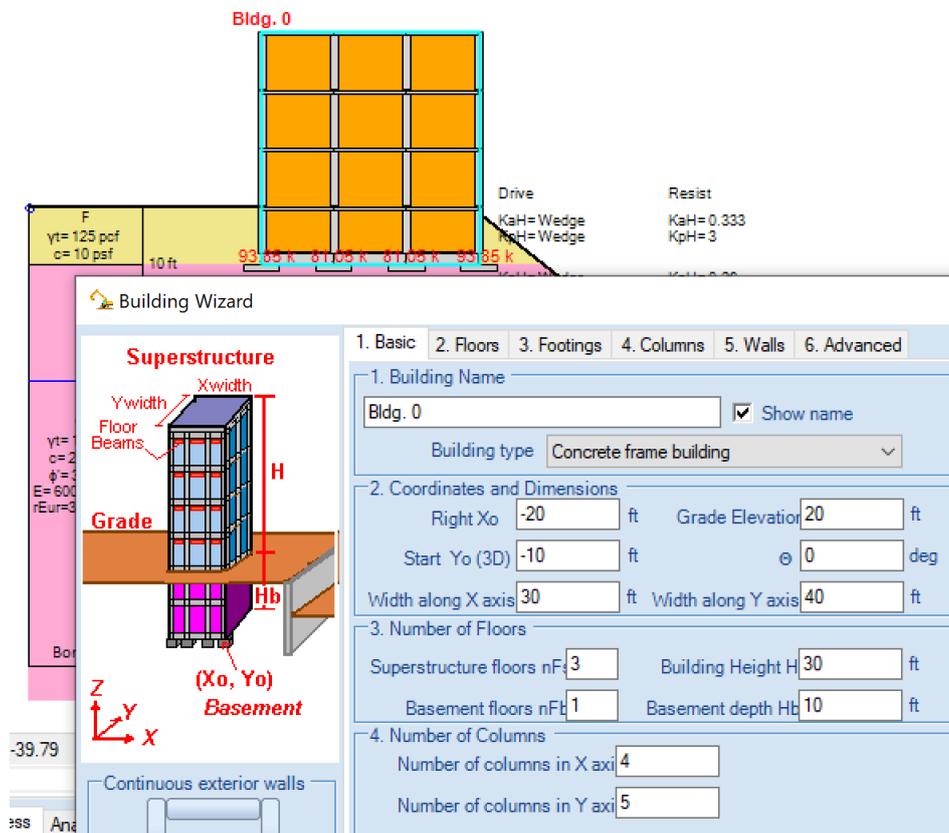


Image: Define Building Properties

Define Analysis Method and Assumptions

In the Analysis tab of the software, we select to use the Finite Element Analysis method for this project. In the Mesh Options dialog, we will select to use the exact theoretical solution for the soil hardening model. In this example we will not use a user-defined convergence tolerance. Finally, we will not use a wall friction in this example.

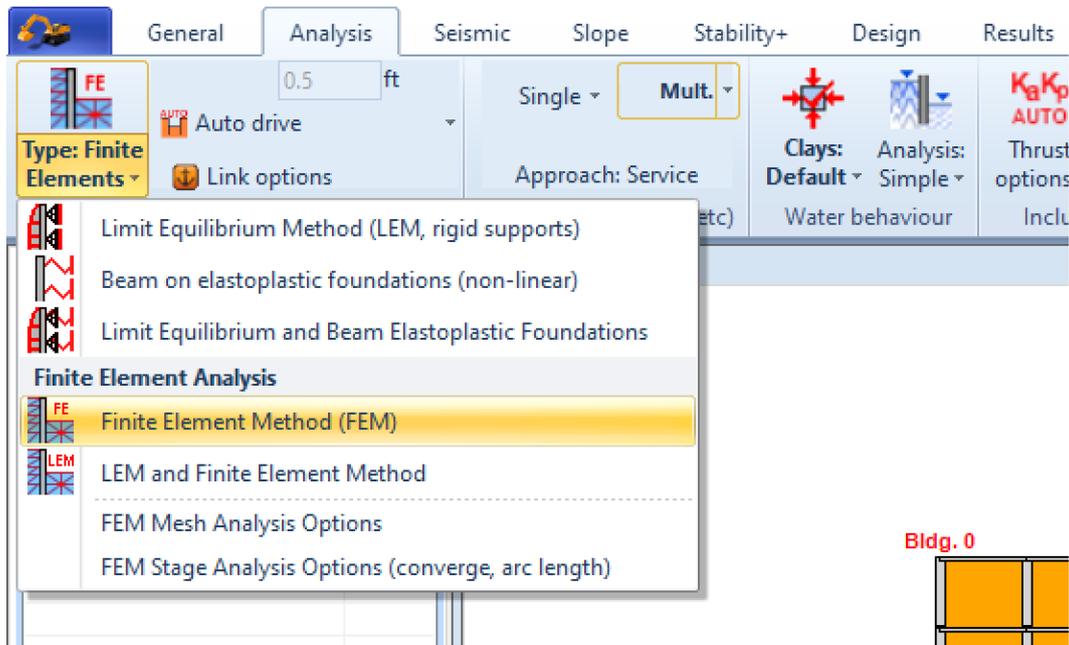


Image: Select Finite Element Analysis Method in DeepEX

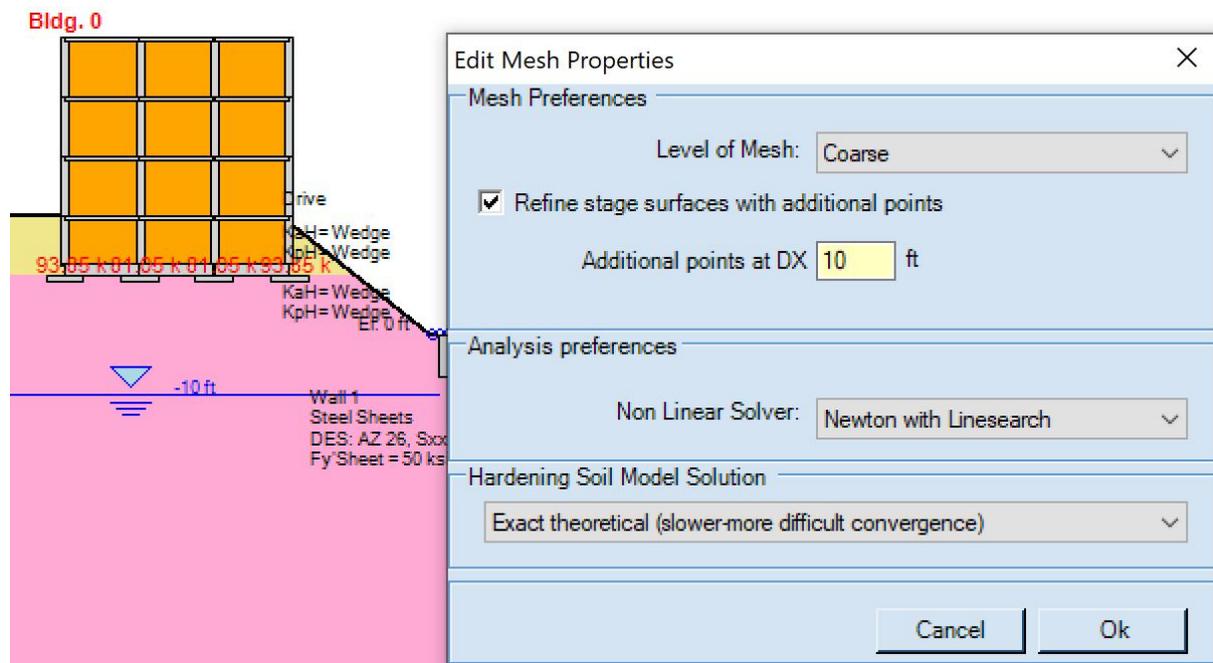


Image: The Mesh Properties dialog – Selected Options

Define Slope Stability Analysis Options

In the Slope tab of DeepEX, we can select to perform slope stability analysis. In this example, we will use a circular slope surface, so we will define a rectangle of possible slope centers close to the slope, and we will also define the surface radius search limits. The slope stability safety factor will be calculated with the General Limit Equilibrium Method (Morgenstern-Price).

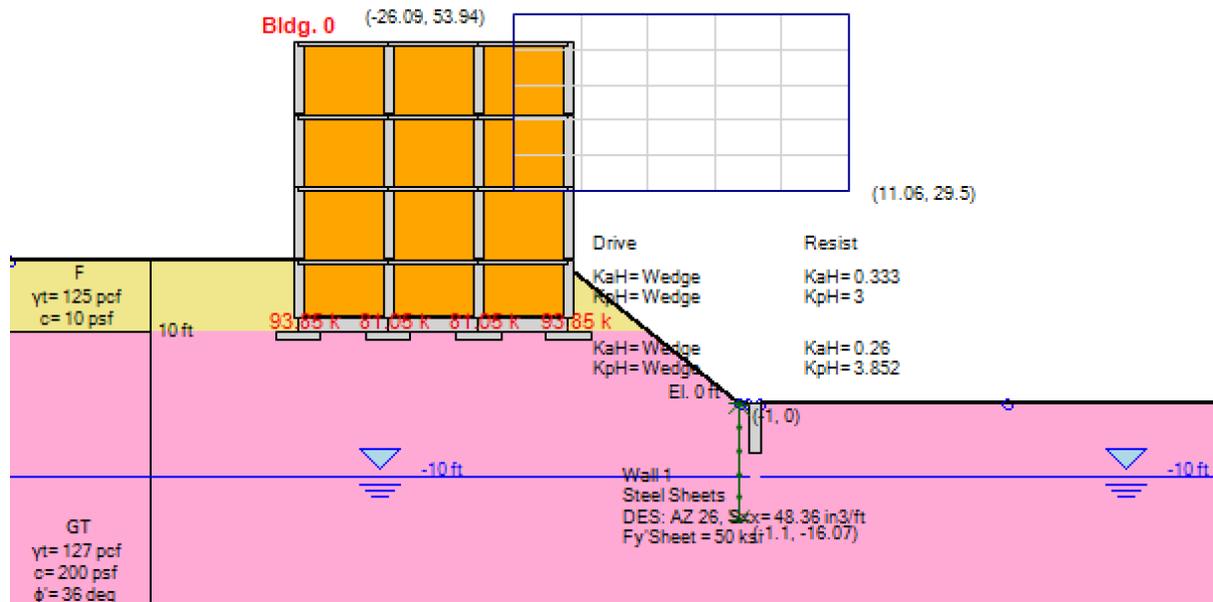


Image: Model in DeepEX – Rectangle of Centers and Radius Search Limits

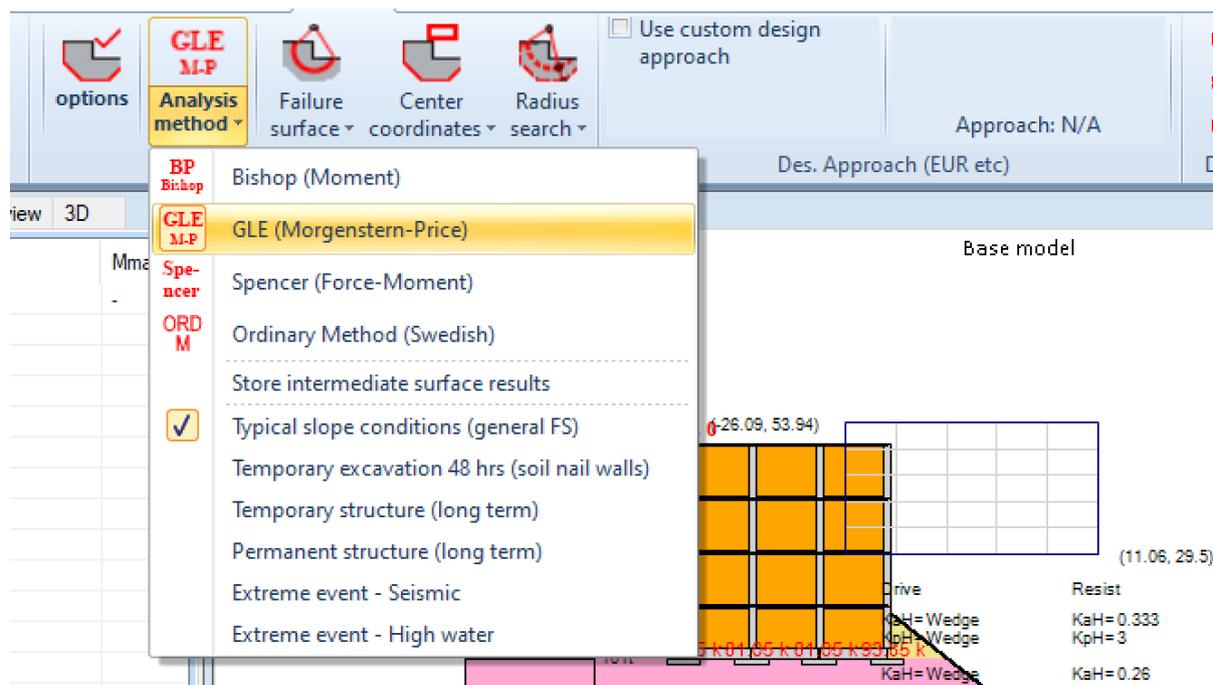


Image: Select Slope Stability Analysis Method – Morgenstern-Price

C. Slope Stability and Finite Element Analysis Results

DeepEX can analyze and present results for all construction stages and created design sections, allowing the designer to review the full project history and take important decisions for optimizing a solution while ensuring adequate safety factors. All results can be presented in Tables and graphically, on the model area. The following images illustrate some of the calculated results, both from the slope stability analysis (critical slope surface and safety factor), and from the finite element analysis (settlements below the building).

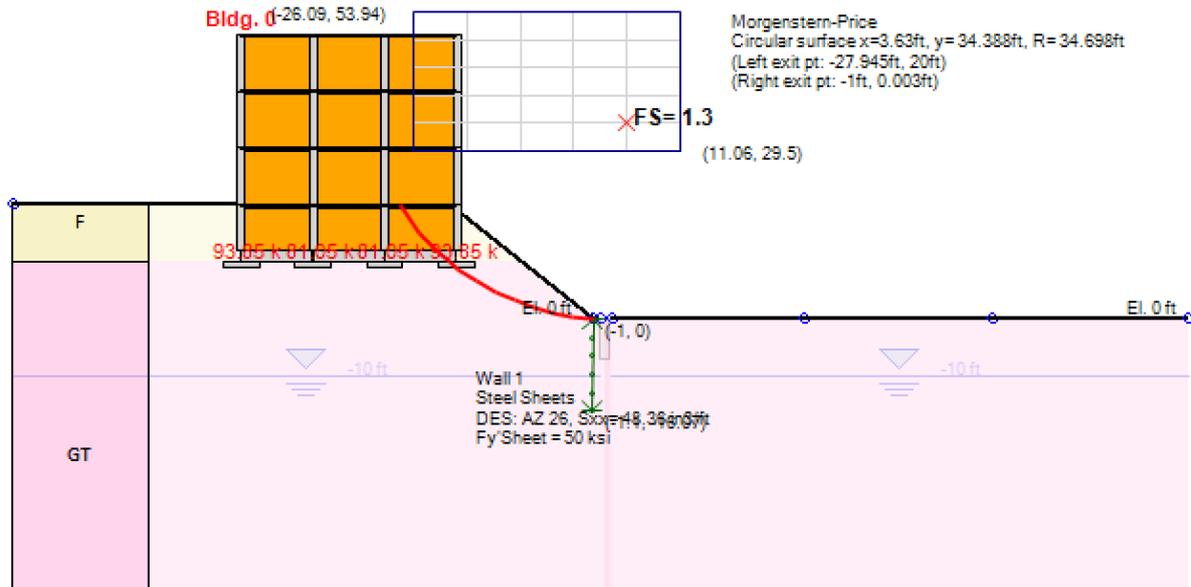


Image: Slope Stability Analysis Results (Critical Slope Surface – Slope Stability FS)

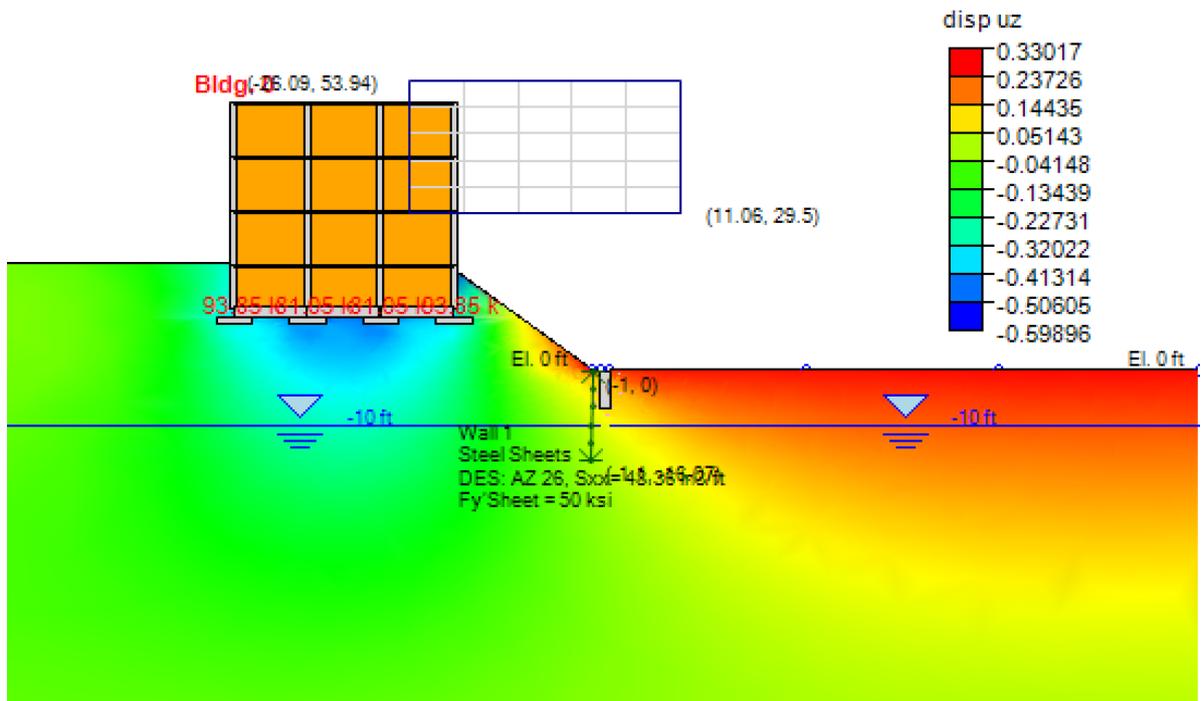


Image: Surface Settlements (FEM Analysis)

12. Example 3: Stepped Anchored Walls – FEM Analysis

A. Introduction – Model Description – Soil Properties – Structural Sections

In this example, we will examine a case of a stepped excavation system (anchored sheet piles), supporting a 60 ft excavation. The water table is at El: -20 ft.

The model will be generated and analyzed with DeepEX – Shoring Design Software, using the Finite Element Analysis method. The following tables present the soil properties, the wall section properties and the support properties respectively.

Table: Soil Properties and Stratigraphy

Layer Top Elevation (ft)	Soil Type	Dry Unit Weight (pcf)	Total Unit Weight (pcf)	Friction Angle (deg)	Cohesion c' (psf)	Soil Model	Modulus of Elasticity (ksf)	Exponent
0	Sand	125	125	30	10	Exponential	300	0.5
-15	Silt	130	130	34	15	Elastoplastic	520	-
-30	Till	135	135	36	300	Elastoplastic	650	-

Table: Wall Properties

Wall	Wall 1 @ X = 0	Wall 2 @ X = 25 ft
Top of the Wall Elevation	0	-30
Wall Depth	40 ft	45 ft
Wall Type	Steel Sheet Piles	Steel Sheet Piles
Wall Section	AZ 26	AZ 26
Steel Material	A50	A50

Table: Support Properties

Sup. Level	Wall	Elevation	Support Type	Support Section	Steel Material
Level 1	Wall 1	-8 ft	Tieback	4 Strands 0.5" @ 10'	Strands 270 ksi
Level 2	Wall 1	-18 ft	Tieback	5 Strands 0.5" @ 10'	Strands 270 ksi
Level 3	Wall 2	-38	Tieback	6 Strands 0.6" @ 10'	Strands 270 ksi
Level 4	Wall 2	-48	Tieback	6 Strands 0.6" @ 10'	Strands 270 ksi

B. Modeling in DeepEX: Construction Stages – Analysis Assumptions – Design Standards

The model will be created graphically, on the DeepEX model area. We will use several construction stages, and in each stage we will perform one step (excavation, support installation), starting from the initial stage 0 (no excavation – at-rest conditions).

Generated Construction Stages

The following image illustrates the model in each construction stage:

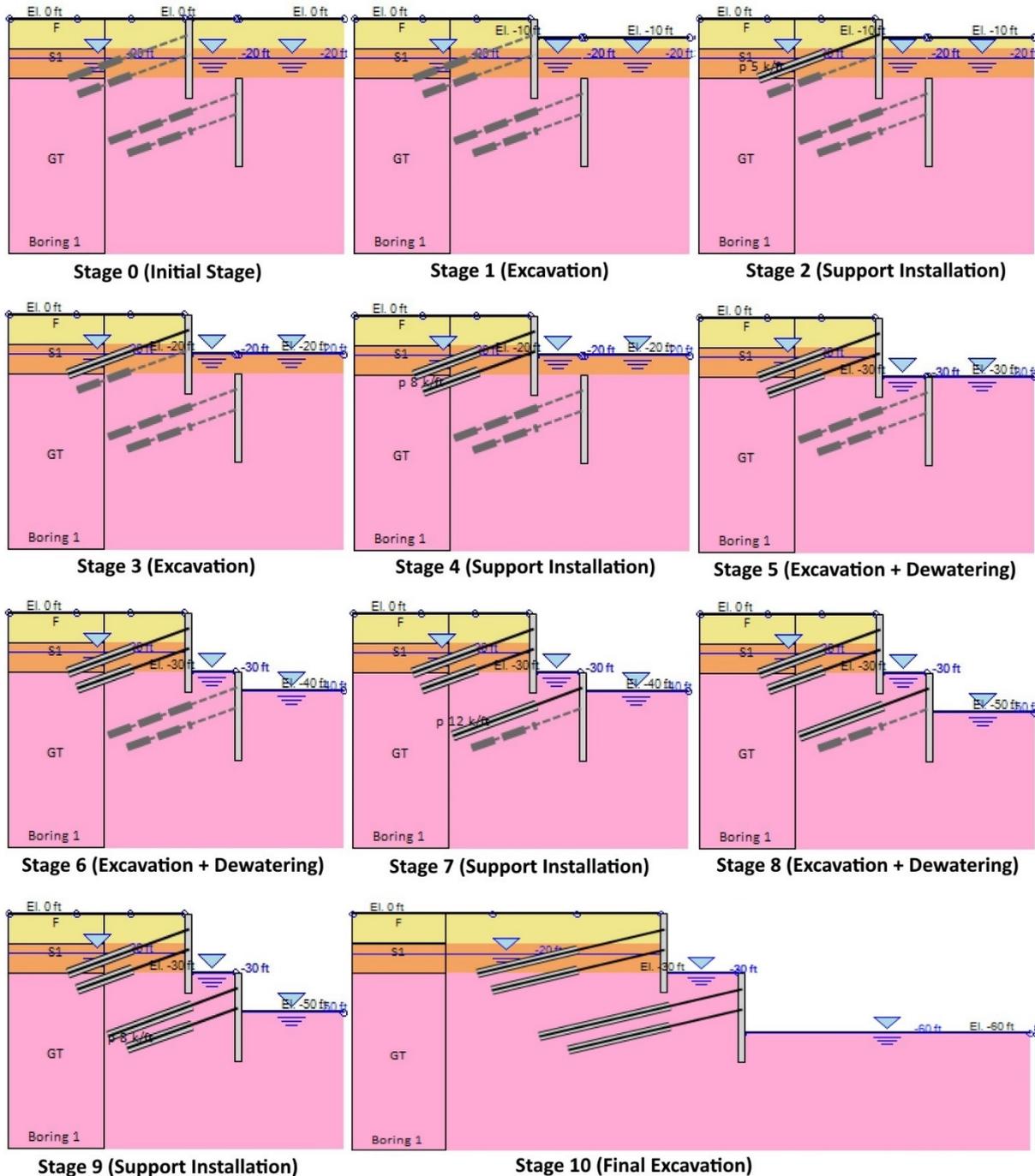


Image: Model in each Construction Stage

Wall Friction and Water Pressures

In this model we will use wall friction on both walls. In the Analysis tab of the software, we select to use on each wall 50% of the soil friction as wall friction.

In this example we will calculate the water pressures using the Simplified Flow analysis method.



Image: Water Pressures method and Defined Wall Friction for each wall

Design Standards

In this model we have used steel sheets and steel struts. In the Design tab of the software, we will select to use AISC 360-16 Allowable method for the calculation of the structural capacities of all steel members.

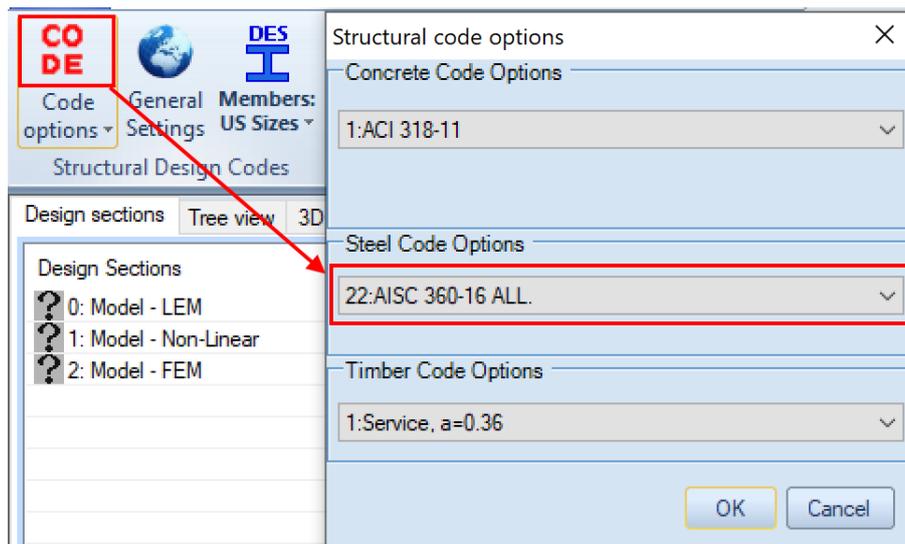


Image: Structural Design Codes

In this example we will perform a Service design, so we will not apply any Geotechnical Standard design combination.

Define Analysis Method and Assumptions

In the Analysis tab of the software, we select to use the Finite Element Analysis method for this project. In the Mesh Options dialog, we will select to use the exact theoretical solution for the soil hardening model. In this example we will not use a user-defined convergence tolerance. Finally, we will not use a wall friction in this example.

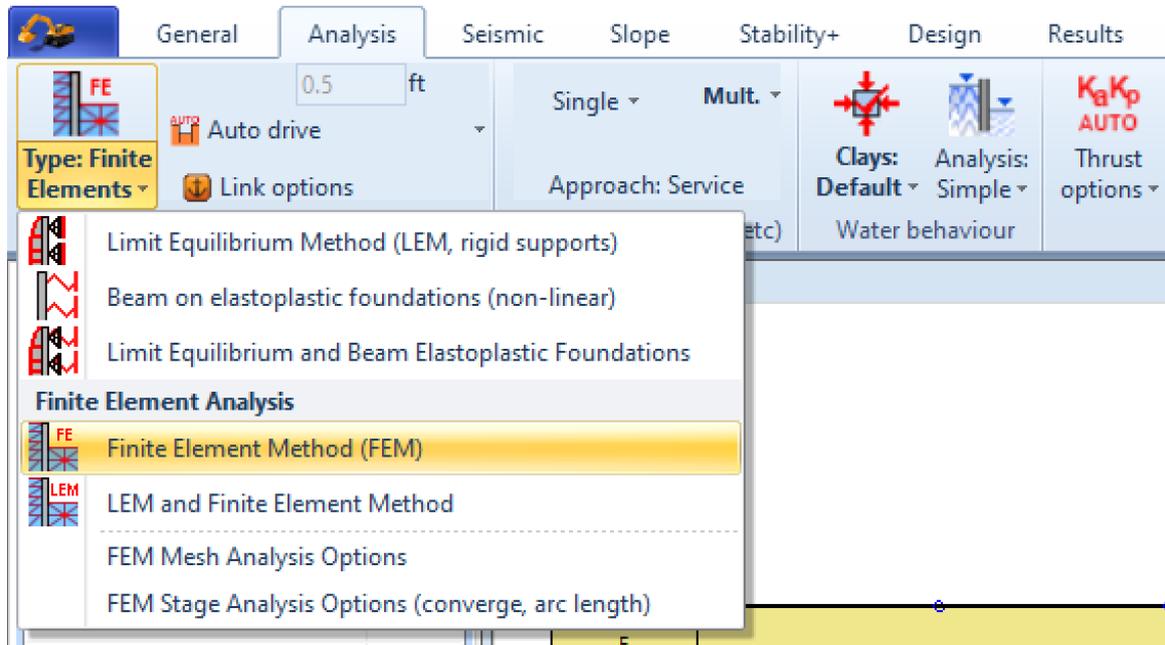


Image: Select Finite Element Analysis Method in DeepEX

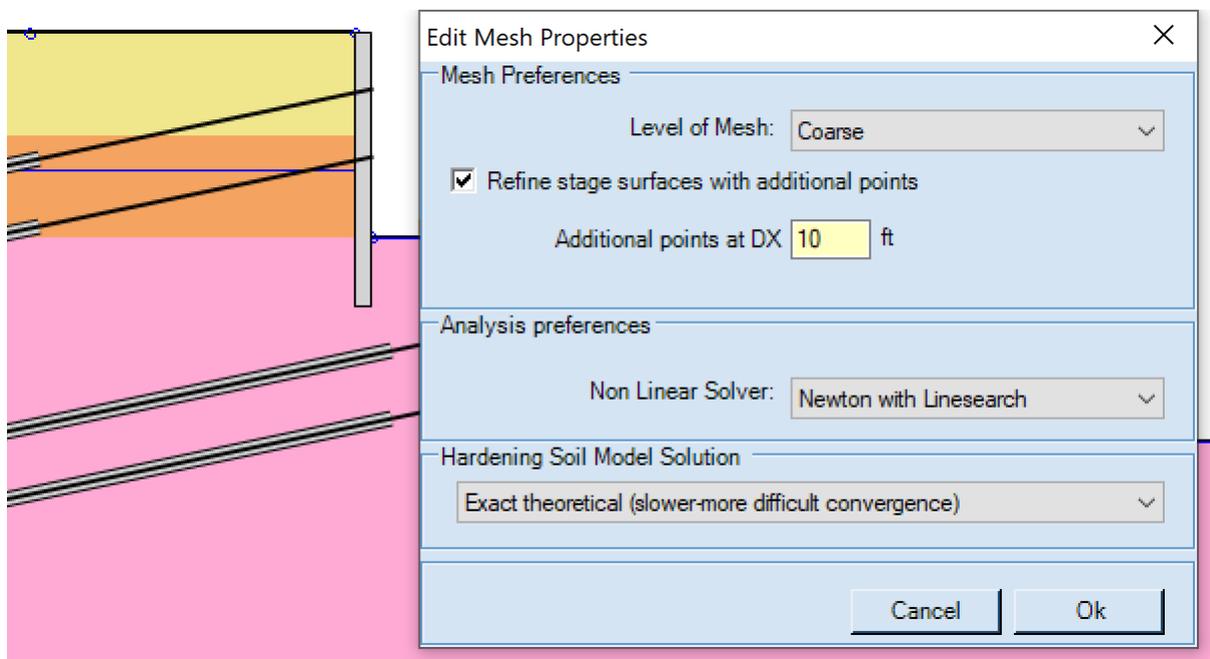


Image: The Mesh Properties dialog – Selected Options

C. Finite Element Analysis Results

DeepEX can analyze and present results for all construction stages and created design sections, allowing the designer to review the full project history and take important decisions for optimizing a solution while ensuring adequate safety factors. All results can be presented in Tables and graphically, on the model area. The following images illustrate some of the calculated results from the finite element analysis (surface settlements, support reactions, wall moment and displacement diagrams).

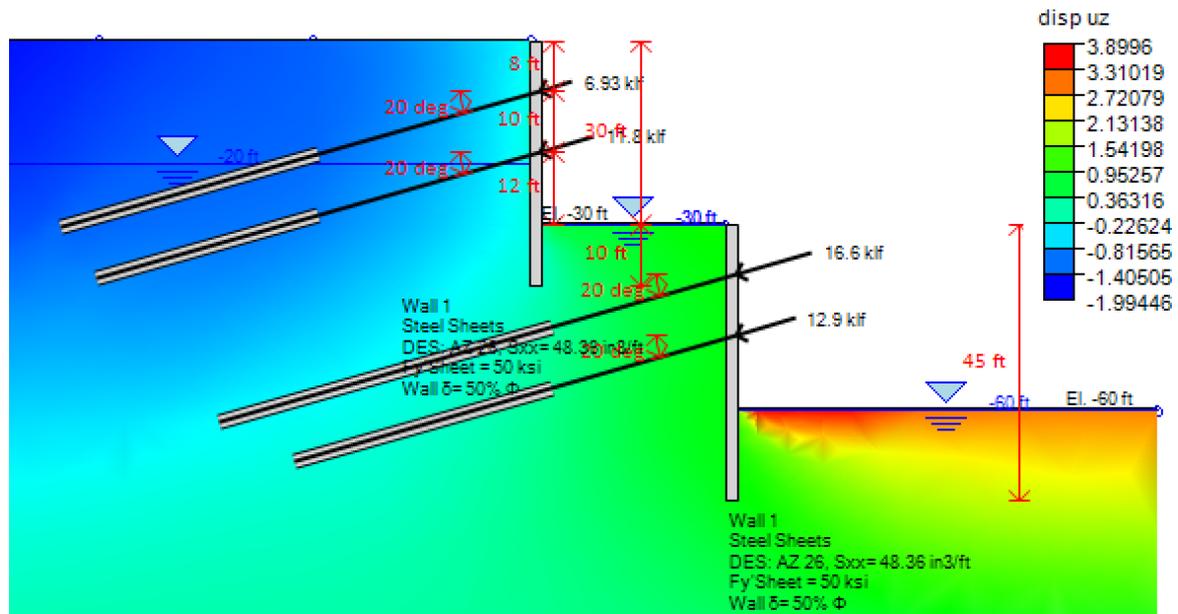


Image: Surface Settlements and Support Reactions (Stage 10)

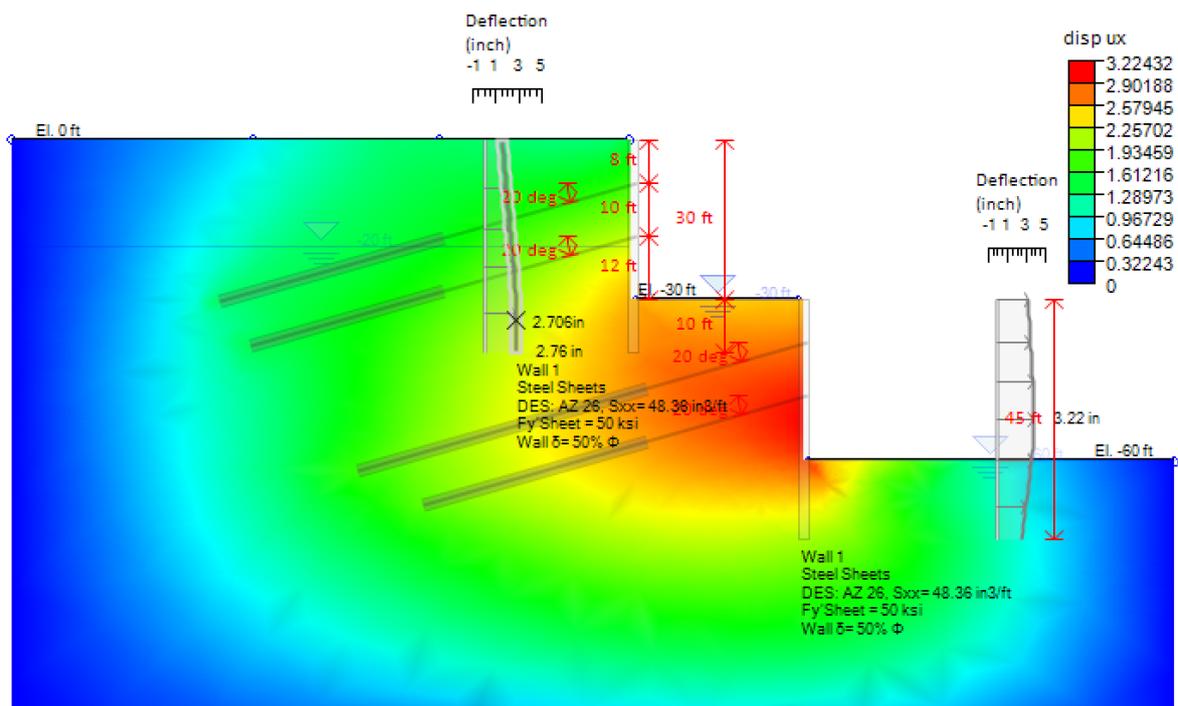


Image: Wall and Soil Displacements (Stage 10)

DEEPFEM – FINITE ELEMENT ANALYSIS IN DEEPEX

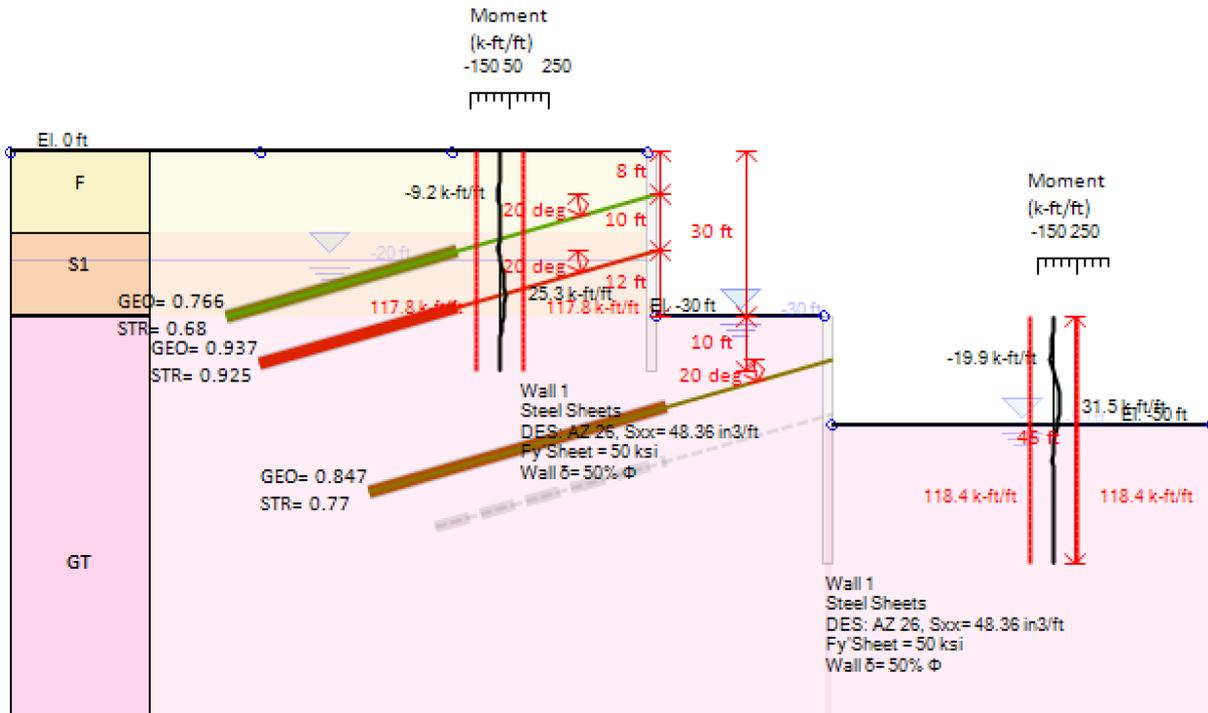


Image: Wall Moments and Support Check Ratios (Stage 8)

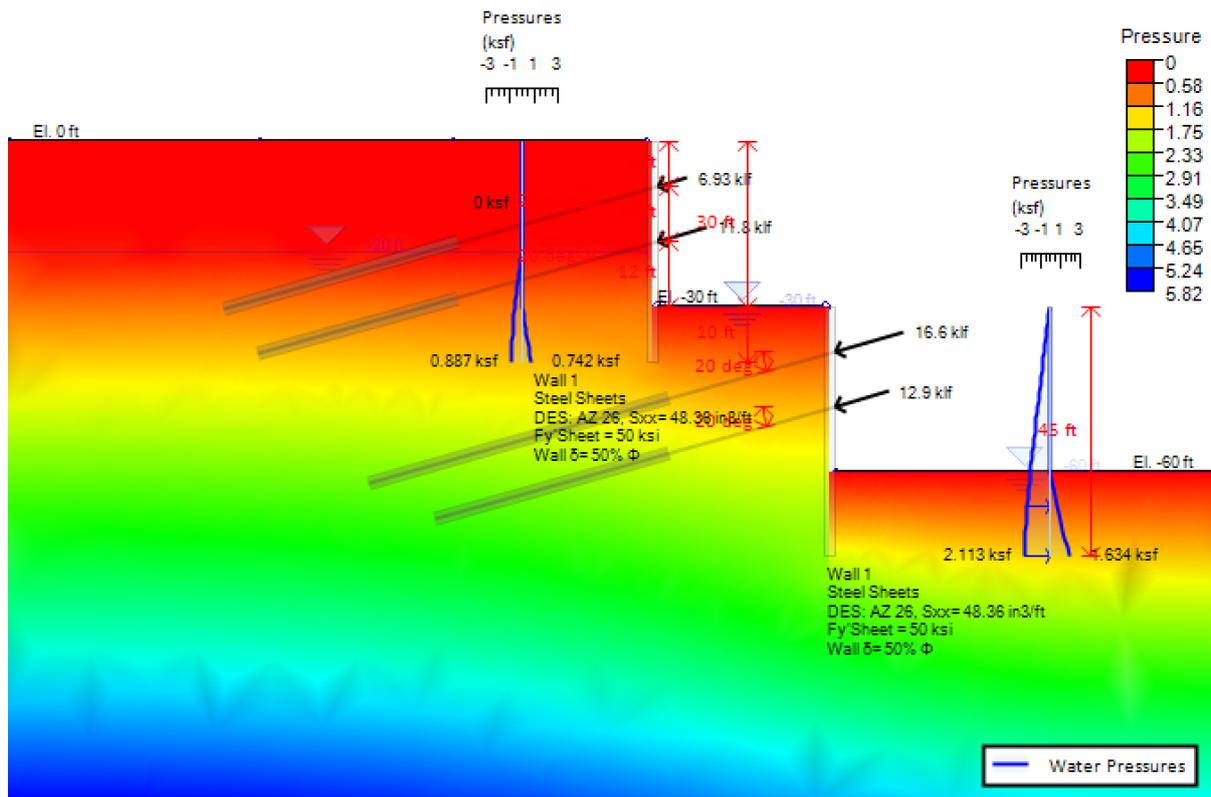


Image: Water Pressures Diagrams and Shadings (Stage 10)