STRUCTURAL CONCRETE SOFTWARE SYSTEM

ADAPT-BUILDER 2019

NEW FEATURES
SUPPLEMENTAL MANUAL

support@adaptsoft.com   www.adaptsoft.com
ADAPT Corporation, Redwood City, California, USA, Tel: +1 (650) 306-2400, Fax: +1 (650) 306-2401
ADAPT International Pvt. Ltd, Kolkata, India, Tel: +91 33 302 86580, Fax: +91 33 224 67281

LIST OF CONTENTS

Chapter 1

INTRODUCTION

1. INTRODUCTION

This supplemental manual provides descriptions and instruction on ADAPT-Builder's latest features. These new features have been introduced to meet the needs for general analysis, design, and reporting capabilities and to increase productivity and efficiency in program use.

These latest features can be categorized into 3 groups: Analysis and Design improvements, Modeling improvements and Reporting improvements.

1.1 Analysis and Design Improvements

- **2nd-order P-Delta Analysis:** The program now includes the option to run 2nd-order analysis for geometric non-linearity ("big" P-Delta) based on the compilation of the elastic and geometric stiffness matrices. A new analysis/design option type for P-Delta is included in which combinations can be singularly run for 2nd-order effects or processed as a batch based on a combination used to obtain the geometric matrix. Graphical analysis results similar to regular combinations can be obtained for P-Delta combinations. These results can be utilized for the design of columns.

- **Drift and Moment Amplification Factors:** With the new P-delta feature, the program graphically reports drift and moment amplification factors (2nd/1st order results) for local axes, RR and SS, as well as the combined global results. The program code checks against a user-defined value set in the Result Browser.

- **Effective Flange for Beams:** The program now includes the option to calculate the effective flange and properties of beams when the support line is set to Beam criteria. The design of the beam sections is performed using the effective properties. The component of stress related to the precompression uses the full tributary area of the section as does the graphical precompression result. ACI and EC2 calculations for effective width are supported.

- **Punching Shear Improvements:** New improvements have been made for punching shear according to ACI318-14. These include:

    - The option to consider critical sections outside the shear reinforced zone with either a rectilinear or octagonal-

    shaped critical section. ACI318-14 Sections 22.6.4.2, 22.6.6.1
  - The option to apply two-way shear provisions for minimum reinforcement for seismic drift. ACI318-14 Sections 18.14.5.1.
  - Application of minimum two-way shear reinforcement at critical sections is based on the requirement at the first section d/2 from face of support.

- **FELT 3D and Long-Term Losses:** When the PT Shop Drawing module is enabled, the program gives new input and functionality for the calculation of long-term losses for unbonded tendons and bonded tendons stressed at the same time. Additional related features include:

  - **FELT 3D Report** – a compiled PDF report summarizing tendon loss calculations including graphical views of the tendons horizontal and vertical profile and the loss diagrams.
  - **Drawing Editor Chair Groups** – Allows the user to input on-demand chair bar maximum spacing and chair bar extensions for graphical output of tendon support bars.

1.2 Modeling Improvements

- **Strip Modeling Dynamic Editor:** A new and improved Strip Modeling Dynamic Editor includes tools used to more rapidly model and modify support lines and design strips. These include:

  - Support Line Wizard – creates a support based on a construction line defined by snap points along the strip path.
  - Support Line Limits – Changes selected support line design criteria and tributary limits for the design strip.
  - Wall – allows the user to set constraints for how walls are considered for design strip generation.
  - Display- Sets the support line display for Direction, Criteria and Width Limit

- **Splitter Functionality:** New and improved splitter functionality. The use of splitters has been simplified to creating a boundary that the tributary edge extends when the width is required to be limited to either side. Splitters are no longer required for any other purpose as the program has been improved to recognize support line nodes at any location and properly generate strips.

- **Ramp Modeling:** Modeling of ramps is now supported in ADAPT-Builder. Ramps are considered analytically only without tendons. Longitudinal and transverse beams can be modeled at ramps with options to automatically offset the beams as well as offset walls and columns supporting ramps. Ramps are required to be modeled in the same plane and the program constrains the graphical input as 3-point input to enforce planar modeling.

- **Beam End Offsets:** Beam coordinate definitions now include the ability to create unique offsets at the beam start and end points necessary to support inclined beams.

1.3 Reporting Improvements

- **Longitudinal Reinforcement on Design Strips:** The Result Browser includes graphical reporting options for design sections to include the area of steel for provided, required and base (user-defined) reinforcement as well as the area of steel/section area (rho).

- **Drift XLS Reporting:** New XLS reporting is available for lateral drift. New reports include list of vertical stations arranged top-down with column coordinates and elevation, X/Y/Global displacements at top and bottom of columns, drift at columns with code check, maximum drift data arranged for easy plot, average story drift, and story drift plot data.

- **FELT 3D Reporting:** A compiled PDF report summarizing tendon loss calculations including graphical views of the tendons horizontal and vertical profile and the loss diagrams.

The following chapters will provide a detailed overview of the new improvements and describe their use and application. Where appropriate, theoretical and/or design code background will be added.

Chapter 2

ANALYSIS AND DESIGN IMPROVEMENTS

2. 2ND-ORDER P-DELTA ANALYSIS

2.1 Defining P-Delta Combinations

A new design/analysis design option type is definable in the Loading Combination input window. Combinations designated as ‘P-Delta’ are processed analytically for 2nd-order effects in one of two ways.

Note that the allowance for analyzing P-Delta combinations is limited to Multi-Level analysis without springs. The program checks for both and will filter out P-Delta combinations in the analysis combination window if either condition is not met.

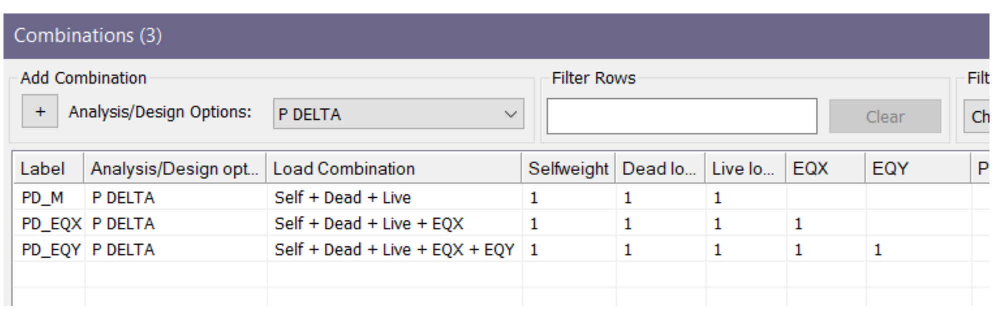
Batch Processing: A geometric stiffness matrix is produced through an iterative procedure based on a master combination. This matrix is combined with a general stiffness matrix for solving subsequent combinations tagged as P-Delta and having identical load cases and factors with additional lateral load cases defined in the combination. An example of this would be the following series of defined combinations:

1.0*SW + 1.0*SDL+ 1.0*LL

1.0*SW + 1.0*SDL + 1.0*LL + 1.0*EQX

1.0*SW + 1.0*SDL + 1.0*LL + 1.0*EQY

In this example, all combinations are defined as ‘P-Delta’ in the Load Combination input window as shown below. Note that naming of combinations for ‘P-Delta’ follows the same rules as those for other combination types. The use of special characters should be avoided, underscore, space or dash can be used as separators and brackets and parentheses are permitted.



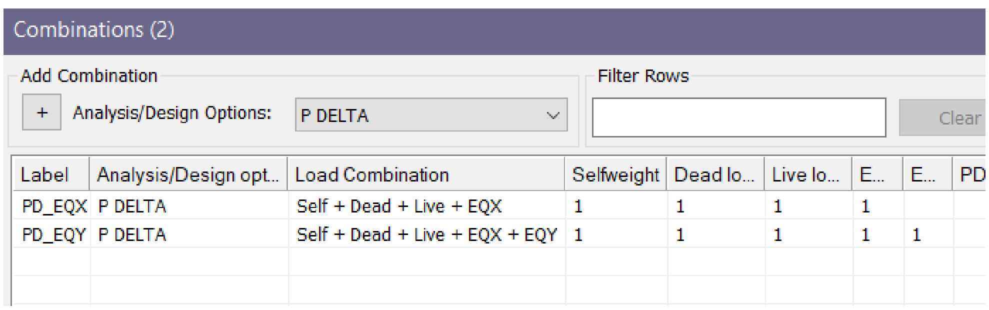
Multiple master combinations are automatically considered if they are set to ‘P-Delta’ and there is at least 1 additional combination that contains the same root set of load cases and factors.

It is recommended and more common to define a master combination that includes only gravity loads or loads that will be sustained on the structure resulting in sustained axial forces in the vertical elements.

Singular Processing: A geometric stiffness matrix is produced through an iterative procedure for each unique ‘P-Delta’ combination. This matrix is combined with a general stiffness matrix for the combination solution. This method requires a significant increase in computation time since each combination requires multiple iterations for convergence. An example of this would be the following series of defined combinations:

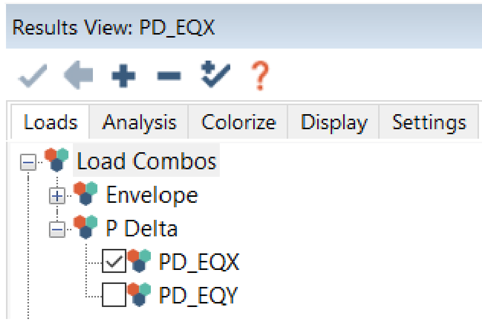
1.0*SW + 1.0*SDL + 1.0*LL + 1.0*EQX

1.0*SW + 1.0*SDL + 1.0*LL + 1.0*EQY



2.2 P-Delta Combination Results

Similar to other combinations defined in the model, combinations set to ‘P-Delta’ that are solved produce general analysis results are selectable through use of the Result Browser. In the ‘Loads’ tab, a new option is shown that lists the solved P-Delta load combinations. If a combination is selected, the ‘Analysis’ tab can be used produce a graphical analysis result.



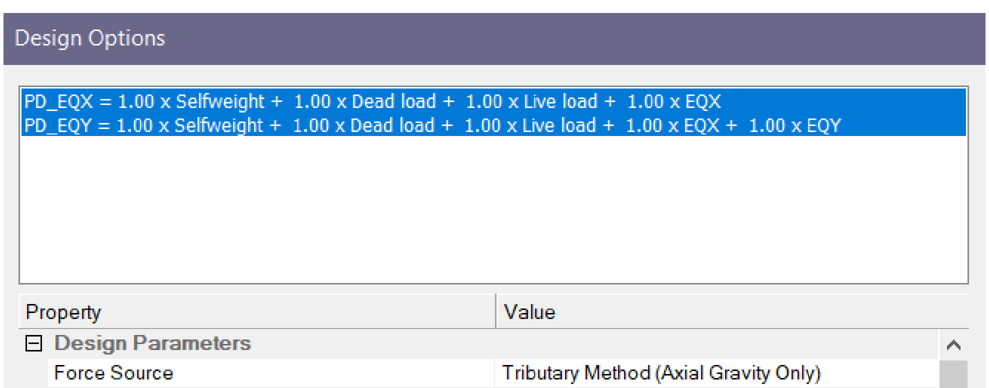
It is advised to create combinations set to NO CODE CHECK in order to obtain direct analysis results comparisons between 1st and 2nd-order effects for the same set of applied loads on the structure. The program does include new Drift and Moment Amplification Factor graphical and tabular results. These are discussed in Section 2.2. General frame results (i.e. moments, shears, displacements) reported graphically in the

Results Browser apply only to the 2nd-order solution for combinations set to the 'P-Delta' option.

The processing and design of both automatic and manual design sections for floor and beam design do not include 2nd-order effects. Combinations with the P-Delta tag are suppressed when designing sections.

Oftentimes, the need arises to design columns in moment frames and other sway structures for the effects of P-Delta. The column design procedure in ADAPT-Builder is allows the user to design columns for any combination solved for in the program, regardless of the analysis/design option type set forth in Loads>Load Combinations.

From Column Design>Column Design Settings the user can include P-Delta combinations in the selection group if a solution for these combinations is available.



2.3 Theoretical Background

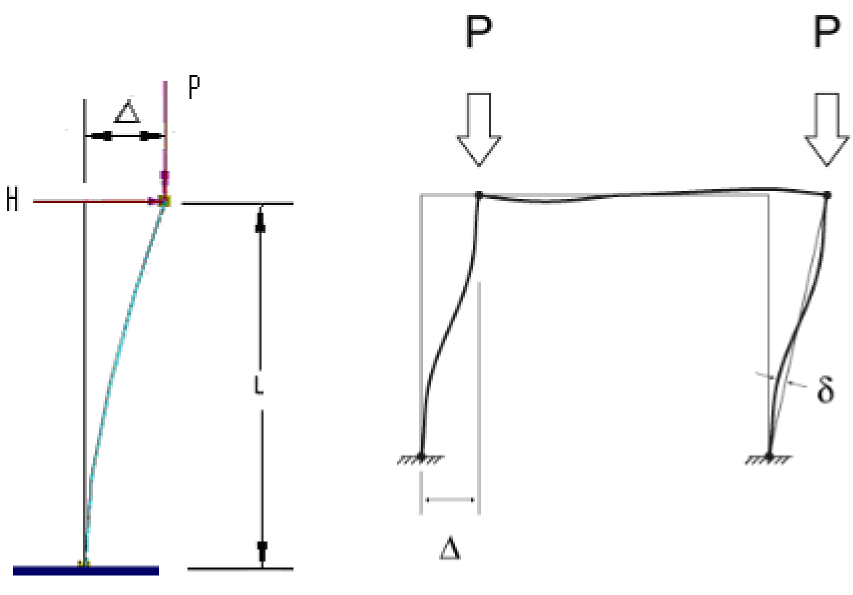
This section summarizes the theoretical formulation and the implementation of PDELTA analysis using ADAPT's FRAME finite element solver.

P-Delta analysis is a methodology to compute structures which are prone to the load-displacement interaction, resulting in second-order effects. This basically means that the load which is applied to the structure is parametrically controlling the elastic or inelastic response of the structure. Such structure cannot be solved using standard (single step) linear elastic process [KS]*[D]=[F], but instead requires the iterative approach.

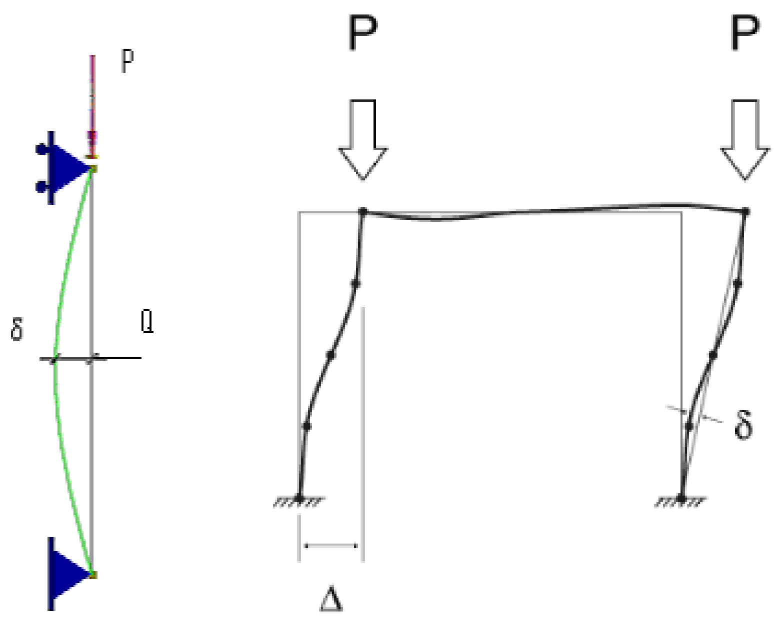
There are two types of P-Delta methodologies:

- Large P-Delta (P-Δ)
- Small P-delta (P-δ)

"Large" P-Delta (P-Δ) refers to the second order effect associated with the lateral translation of the members. The idea of large P-Delta (P-Δ) is illustrated in the following diagrams:



"Small" P-delta (P-δ) refers to the second order effect associated with the member curvature. The idea of small P-delta (P-δ) is illustrated in the following diagrams:



The difference between these two methodologies lies in the mathematical formulation as well as in FE solution approach. On the level of mathematical formulation (which translates into FE formulation) the difference can rely in excluding or including higher order terms (geometric nonlinearity).

On the level of a finite element (FE) solution, the difference between these two methodologies will also depend on the granularity of meshing as well as prescribed number of iterations. For refined meshes, the solutions using both approaches may lead to similar solutions.

The most accurate methodology is when nonlinear 2nd-order FE analysis is used. In such case the effects of member curvature are covered by high-order formulation (geometric nonlinearity), and for this reason it is a (small) P-delta solution.

Linear elastic second order analysis is commonly used in engineering practice and allowed by design codes (e.g. ACI 318). Linear elastic 2nd-order analysis which is based on geometric stiffness matrix is considered sufficiently accurate for engineering applications.

The implementation focus in ADAPT-Builder's frame solver is that of P-Delta (P-Δ) 2nd-order, linear elastic, and small displacements calculations in conjunction of geometric stiffness matrix of frame (beam/column) element.

2.3.1 Definition of Geometric Stiffness

A geometric stiffness matrix K_G accounts for the effect of loads existing in the element on the stiffness of the element. For example, it is well known that the axial load in a beam-column has an appreciable effect on the lateral stiffness. The geometric stiffness matrix K_G is an adjustment to the conventional elastic stiffness matrix to account for such effects.

The adjusted (combined) stiffness matrix of the entire system is used in the course of finite element computation, to obtain a solution which incorporates the effects of load on the stiffness of the system. Such computational process requires iterative or a pseudo-iterative approach in order to converge to a solution for which the adjusted (combined) stiffness matrix corresponds well to the applied loads, and the displacements for two consecutive iterations are within prescribed tolerance.

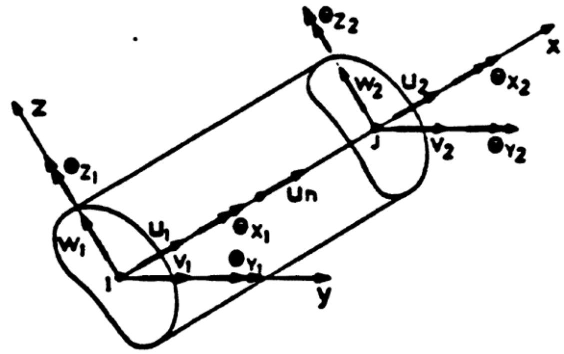
The geometric stiffness matrix can be derived for frame (beam and column) elements as well as for plate shell elements. The derivation of the geometric stiffness matrix for frame elements is relatively simple and straight-forward.

The derivation of geometric stiffness for plate triangular and quadrilateral elements is much more complex, and it presents multiple mathematical and numerical challenges. The current implementation in ADAPT-Builder’s frame solver does not cover the geometric stiffness of plate shell element.

The focus for the current implementation is on geometric stiffness of frame element, since it plays a prominent role, when computing structures supported on columns such as unbraced or partially braced multi-level frames and buildings.

2.3.2 Formulation of Elastic and Geometric Stiffness for Frame Element

A standard 3-dimensional (Bernoulli-Euler) frame element is assumed, having the following local coordinate system:



The elastic stiffness matrix of such a 3-dimensional frame element in local coordinates system can be represented by the following 12x12 matrix:

$$\bar{\mathbf{k}} = \frac{E}{L}\begin{bmatrix}
A & 0 & 0 & 0 & 0 & 0 & -A & 0 & 0 & 0 & 0 & 0 \\
0 & 12I_z/L^2 & 0 & 0 & 0 & 6I_z/L & 0 & 0 & -12I_z/L^2 & 0 & 0 & 6I_z/L \\
0 & 0 & 12I_y/L^2 & 0 & -6I_y/L & 0 & 0 & 0 & -12I_y/L^2 & 0 & -6I_y/L & 0 \\
0 & 0 & 0 & J/2(1+v) & 0 & 0 & 0 & 0 & 0 & -J/2(1+v) & 0 & 0 \\
0 & 0 & -6I_y/L & 0 & 4I_y & 0 & 0 & 0 & -6I_y/L & 0 & 2I_y & 0 \\
0 & 6I_z/L & 0 & 0 & 0 & 4I_z & 0 & -6I_z/L & 0 & 0 & 0 & 2I_z \\
-A & 0 & 0 & 0 & 0 & 0 & A & 0 & 0 & 0 & 0 & 0 \\
0 & -12I_z/L^2 & 0 & 0 & 0 & -6I_z/L & 0 & 12I_y/L^2 & 0 & 0 & 0 & -6I_z/L \\
0 & 0 & -12I_y/L^2 & 0 & 6I_y/L & 0 & 0 & 0 & 12I_y/L^2 & 0 & 6I_y/L & 0 \\
0 & 0 & 0 & -J/2(1+v) & 0 & 0 & 0 & 0 & 0 & J/2(1+v) & 0 & 0 \\
0 & 0 & -6I_y/L & 0 & 2I_y & 0 & 0 & 0 & 6I_y/L & 0 & 4I_y & 0 \\
0 & 6I_z/L & 0 & 0 & 0 & 2I_z & 0 & -6I_z/L & 0 & 0 & 0 & 4I_z
\end{bmatrix}$$

The corresponding geometric stiffness matrix of 3-dimensional frame element in local coordinates system can be represented by the following 12x12 matrix:

$$\bar{k}_g = \frac{P}{L}\begin{bmatrix}
1 & 0 & 0 & 0 & 0 & 0 & -1 & 0 & 0 & 0 & 0 & 0 \\
0 & \frac{6}{5} & 0 & 0 & 0 & \frac{L}{10} & 0 & -\frac{6}{5} & 0 & 0 & 0 & \frac{L}{10} \\
0 & 0 & \frac{6}{5} & 0 & -\frac{L}{10} & 0 & 0 & 0 & -\frac{6}{5} & 0 & -\frac{L}{10} & 0 \\
0 & 0 & 0 & \frac{I_\rho}{A} & 0 & 0 & 0 & 0 & 0 & -\frac{I_\rho}{A} & 0 & 0 \\
0 & 0 & -\frac{L}{10} & 0 & \frac{2L^2}{15} & 0 & 0 & 0 & \frac{L}{10} & 0 & -\frac{L^2}{30} & 0 \\
0 & \frac{L}{10} & 0 & 0 & 0 & \frac{2L^2}{15} & 0 & -\frac{L}{10} & 0 & 0 & 0 & -\frac{L^2}{30} \\
-1 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 \\
0 & -\frac{6}{5} & 0 & 0 & 0 & -\frac{L}{10} & 0 & \frac{6}{5} & 0 & 0 & 0 & -\frac{L}{10} \\
0 & 0 & -\frac{6}{5} & 0 & \frac{L}{10} & 0 & 0 & 0 & \frac{6}{5} & 0 & \frac{L}{10} & 0 \\
0 & 0 & 0 & -\frac{I_\rho}{A} & 0 & 0 & 0 & 0 & 0 & \frac{I_\rho}{A} & 0 & 0 \\
0 & 0 & -\frac{L}{10} & 0 & -\frac{L^2}{30} & 0 & 0 & 0 & \frac{L}{10} & 0 & \frac{2L^2}{15} & 0 \\
0 & \frac{L}{10} & 0 & 0 & 0 & -\frac{L^2}{30} & 0 & -\frac{L}{10} & 0 & 0 & 0 & \frac{2L^2}{15}
\end{bmatrix}$$

Where:

P – axial force in the element
L – length of the element
$I\rho$ – Ix+Iy – Polar moment of inertia

The combined elastic+geometric stiffness matrix of frame element as implemented in ADAPT's Frame can be represented by the sum of each matrix location entry in from the matrices shown above.

2.3.3 Description of the Solution Process

The combined elastic and geometric stiffness matrices [Ke+Kg] of individual frame elements are transformed from the local to global coordinate system and assembled into a stiffness matrix of the entire system [K_s]. The results are obtained in a similar fashion by solving the system of finite element equations:

$$[K_s]*[D]=[F]$$

Where:

[D] – nodal displacement vector (unknown)
[F] – nodal force (load) vector.

Because geometric matrices [K_g] are parametrized by the axial forces acting in the elements, the solution needs to be calculated iteratively, and the elements of geometric matrices updated for each iteration. In the first iteration, the system is solved as a regular elastic model, without use of the geometric stiffness matrices. In the subsequent solutions, geometric matrices are employed. Typically, iterative processing converges quickly, especially when analyzing multi-level structures, where the magnitude of axial forces in the elements does not change significantly from one iteration to the other. For practical purposes 2 or 3 iterations are sufficient to obtain a relevant solution.

Because P-Delta analysis is a pseudo-nonlinear process, the solutions should normally be obtained using complete load combinations instead of individual load cases, since the solutions cannot be linearly combined. This causes the amount of time needed to obtain the solutions for multiple combinations to be more significant and computationally consumptive than when solving individual load cases.

In typical engineering practice, P-Delta analysis is employed in conjunction with lateral analysis, when wind or earthquake loads are acting. The purpose is to capture the amplification of the lateral effects due to P-Delta effects. For such cases, literature suggests that simplifications are possible in the process, so as to limit the computation time. For example, when the following load combination is involved:

$$U=1.2D + 1.0L + 0.2S + 1.0E$$

there are typically several variations of it, because the seismic load can be acting in a few different ways. Literature sources suggest that the amplification of lateral loads due to P-Delta effects may, in such situations, be obtained by evaluating the geometric stiffness matrices based on the vertical and sustained gravity components of the load combination such as:

$$U=1.2D + 1.0L + 0.2S$$

The assumption here is that the lateral loads have negligible effect on axial forces in vertical elements (columns). Under this assumption several combinations of the same type can be solved in parallel using the same iteration process. The first “master” load case (or vertical combination) may be solved, such as:

$$U=1.2D + 1.0L + 0.2S$$

This combination is used to evaluate the geometric matrices. The remaining combinations, such as all the lateral variations of:

$$U=1.2D + 1.0L + 0.2S + 1.0E$$

can be solved in parallel using the same set of system matrices. This performance improvement is implemented in FRAME as part of batch solutions as described later in the document.

2.3.4 Prestressing in Conjunction with P-Delta

There is limited information with regards to how prestressing or post-tensioning forces are to be handled when performing P-DELTA analysis. The comments contained in this section are for the most part the opinion of the ADAPT and may not be in agreement with other literature sources.

Prestressing is a source of compressive axial forces in the elements. Prestressing and post-tensioning tendons are embedded inside the element (if not external), therefore eccentricity of the tendon remains unchanged, with regards to the deformed section. Subsequently there is no amplification effect for the tendon's transverse actions as a result of the compressive prestressing force. Similarly, there is no amplification effect when a prestressing force acts in conjunction with transverse loads different than PT. An embedded tendon itself is not a source of buckling (secondary) effect for the host concrete element.

For this reason, it is unnecessary to account for the axial force due to prestressing, when evaluating the elements of the geometric stiffness matrices. The axial forces to be used for evaluating the geometric stiffness matrices should primarily be based on gravity load components. The transverse effects of the post-tensioning ("load balancing" effects) may interact with the axial forces in the elements due to external loads, such as gravity loads, leading to the secondary (P-Delta) effects. In other words, the P-Delta effects are possible, when tendon exerts transverse load on the element in conjunction with external (non-PT) axial force within the element. The reason for this is that post-tensioning generates the deformation of the member, therefore an additional external axial force acting on the member may lead to the amplification of the deformation caused by the tendon.

Let us assume that we have the following load combination to be analyzed for P-Delta:

U=1.2D + 1.0L + 0.2S + 1.0P + 1.0E

Based on what was written before in this section, two load cases in this combination, prestressing "P" and lateral "E", are the obvious candidates to be excluded from evaluation of the geometric stiffness matrices.

The implementation of P-Delta in ADAPT-Builder's frame solver automatically excludes the contribution of prestressing force from the calculation of geometric stiffness matrix. This takes place for all possible arrangements of load cases (load combinations) in the INP input file, both in singular and batch processing modes.

2.3.5 P-Delta Analysis Using Load Batches

ADAPT-Builder's frame solver has the capability to solve multiple load cases in single solution handling (using the same system stiffness matrix). In such a case, the right-hand side of the FEM equation system consists of multiple nodal load vectors, and the solution consists of multiple vectors of nodal displacements. Normally these types of solutions are only possible for linear analysis. The advantage of this feature is a significant reduction of the computation time, because the system stiffness matrix needs to be solved only once for multiple load combinations.

Since P-Delta is pseudo nonlinear, it is not possible to obtain fully accurate P-Delta solutions when using load batches. When analysis is performed using load batches, the first load set (partial combination) from the batch will be used to evaluate the combined stiffness matrix of the system $[K_e+K_g]$, while the remaining load sets in the batch may include the load components which are negligible or inappropriate for calculation of the geometric stiffness.

For the load combination mentioned above, the first load set (partial combination, or "master" combination) to be used for evaluating the combined stiffness matrix of the system $[K_e+K_g]$ will be as follows:

U=1.2D + 1.0L + 0.2S + 1.0P

In the above combination, the prestressing component will be ignored by the program, when evaluating geometric stiffness matrices, but will be considered otherwise.

The remaining load sets for given load batch may represent the complete combinations such as:

U=1.2D + 1.0L + 0.2S + 1.0P + 1.0E

Typically, there are several variations of the lateral load "E" (or similarly, "W" for wind), therefore the load batch will produce several solutions, using single system matrix [K_e+K_g] if those variations are included as combinations.

Similar multi-load solutions can be obtained when the lateral load is of wind type.

The usage of batch loads required additional provisions in ADAPT-Builder's GUI, in order to automate the process of generation of INP input file. The algorithm developed in the GUI allows for combinations to be set to the 'P-Delta" analysis/design option type. This algorithm performs the following functions:

1. Load Combinations will be internally sorted according to:

    - combination category (SLS, ULS etc)
    - type of analysis (P-DELTA included or excluded)
    - combination factors for each load component (SW, DL, SDL, LL, PT EQ etc)

2. Load combinations which are selected for P-Delta analysis will be assigned into groups (batches) based on identical load factors for vertical (gravity) load factors.

    The group (batch) which will be solved together needs to have all load components and factors identical, except for the lateral component.

3. For each P-Delta load combination batch, the algorithm creates a master combination which includes identical load components. This master load combination is used to establish the geometric matrix.

The algorithm for P-Delta load combination post-processing, has the ability to switch from batch load solutions to individual solutions. Although the load batch approach is oftentimes sufficiently accurate when analyzing multi-floor buildings with prevailing gravity loads, it may sometimes be insufficient. The program gives the user an option to switch from batch load solutions to individual solutions.

Individual solutions may be computationally more consuming, but sometimes can be useful for the engineer. The use of individual solutions is especially justified, when structural model is fully developed and requires final results.

If P-Delta load combinations are developed such that there is no unique set of load cases and factors that is repeated, each combination is solved independently with a unique geometric matrix, not shared.

2.3.6 Special Considerations

In ADAPT-Builder’s frame solver, the computations using P-Delta methodology can only be performed for frame (building) structures. Currently the frame solver does not support P-Delta computations for models which are supported on uni-directional (compression only) springs. The use of uni-directional springs requires an iterative (pseudo-nonlinear) solution, which would be in conflict with the iteration process for P-Delta analysis as currently implemented. ADAPT-Builder’s GUI performs a check for uni-directional springs, and if present, disables the option to run P-Delta analysis. In particular P-Delta analysis is disabled, when analyzing models in MAT or SOG modes of the program.

The FEM analysis which includes geometric stiffness matrices may on occasions result in non-solvable models. This is possible when the magnitude of the axial loads in the columns is close to or above the critical buckling loads. Even if for a single column, the value of axial load exceeds critical, the resulting combined stiffness matrix $[K_e+K_g]$ of this element may be singular and unsolvable. It is advisable that an engineer should perform preliminary static (non-P-delta) analysis first, in order to evaluate the magnitude of axial forces in columns, to make sure their critical loads are not exceeded. Graphical reporting of amplification factors for both moment and drift are available for P-delta load combinations.

2.3.7 Selected ACI Design Code Provisions Related to P-Delta Analysis

ACI-318 code defines the requirements for structural analysis of concrete member design and permits the 1st order analysis with moment magnification, and/or, elastic second 2nd analysis. Other methods included are inelastic 2nd-order and non-linear FEM.

Stability Limit

Starting from the 2008 edition of ACI 318 code, a unified stability requirement was introduced on compressive members (primarily columns) which limits the moment amplification due to P-Delta to 1.4 as a ratio of 2nd/1st order moments. This provision was presented in this edition of the code as follows:

10.10.2.1 — Total moment including second-order effects in compression members, restraining beams, or other structural members shall not exceed 1.4 times the moment due to first-order effects.

R10.10.2.1 — If the weight of a structure is high in proportion to its lateral stiffness, excessive *PΔ* effects (where secondary moments are more than 25 percent of the primary moments) may result, which will eventually introduce singularities into the solution to the equations of equilibrium, indicating physical structural instability.[10.34] Analytical research[10.35] on reinforced concrete frames showed that the probability of stability failure increases rapidly when the stability index Q exceeds 0.2, which is equivalent to a secondary-to-primary moment ratio of 1.25. According to ASCE/SEI 7-05,[10.36] the maximum value of the stability coefficient θ, which is close to the ACI stability coefficient Q, is 0.25. This value is equivalent to a secondary-to-primary moment ratio of 1.33. The upper limit of 1.4 on the secondary-to-primary moment ratio was chosen considering the above. By providing an upper limit on the second-order moment, it is unnecessary to retain the stability check given in 10.13.6 of the 2005 Code.

When structural model satifies this requirement for each of its columns, its stability is is assured both globally and locally.

The funtionality for P-Delta analysis inside ADAPT-Builder's frame solver includes new procedures for calculation of the following amplification factors:

- MAF – moment amplification factors at the ends of frame elements.
- DAF – drift amplification factors.

These values are stored in ADO output files for P-Delta solutions and are used as the source files for producing both graphical and XLS output for moment, drift and displacements.

In the previous editions of the ACI code, the basic measure of stability was based on the so-called “stability index.” ADAPT-Builder’s frame engine does not support this particular calculation and stability is reported in the aforementioned method.

Section Stiffness Reduction

The 2008 edition of the ACI318 code sets forth the following requirements regarding elastic 2nd-order P-Delta analysis:

> **10.10.4 — Elastic second-order analysis**
>
> Elastic second-order analysis shall consider section properties determined taking into account the influence of axial loads, the presence of cracked regions along the length of the member, and the effects of load duration.

In lieu of more precise determination of the properties of cracked sections, the following simplified stiffness reductions are permissible:

10.10.4.1 — It shall be permitted to use the following properties for the members in the structure:

(a) Modulus of elasticity..................... E_c from 8.5.1
(b) Moments of inertia, I
  Compression members:
    Columns .... $0.70I_g$
    Walls—Uncracked .... $0.70I_g$
      —Cracked....................................... $0.35I_g$
  Flexural members:
    Beams .... $0.35I_g$
    Flat plates and flat slabs.......................... $0.25I_g$
(c) Area... $1.0A_g$

Alternatively, the moments of inertia of compression and flexural members, I, shall be permitted to be computed as follows:

Compression members:

$$I = \left(0.80 + 25\frac{A_{st}}{A_g}\right)\left(1 - \frac{M_u}{P_u h} - 0.5\frac{P_u}{P_o}\right)I_g \le 0.875I_g \qquad (10\text{-}8)$$

where P_u and M_u shall be determined from the particular load combination under consideration, or the combination of P_u and M_u resulting in the smallest value of I. I need not be taken less than $0.35I_g$.

Flexural members:

$$I = (0.10 + 25\rho)\left(1.2 - 0.2\frac{b_w}{d}\right)I_g \le 0.5I_g \qquad (10\text{-}9)$$

The above stiffness reductions are primary prescribed for non-prestressed members. For prestressed members ACI318 states the following:

The values of the moments of inertia were derived for nonprestressed members. For prestressed members, the moments of inertia may differ depending on the amount, location, and type of the reinforcement and the degree of cracking prior to ultimate. The stiffness values for prestressed concrete members should include an allowance for the variability of the stiffnesses.

Assuming that structural floors and beams are prestressed, the reductions of stiffness included in section 10.10.4.1 are at least applicable to columns and walls. This means that PT structural models should, at the very minimum, be executed using flexural stiffness reduction factors equal to 0.7 for all vertical support elements (columns and walls). In case it is hard to determine the state cracking in structural walls, the more appropriate flexural stiffness reduction factor equals 0.35.

ADAPT-Builder allows users to define 'Usage Cases' to set modifiers for the structural components. A user is able to define, store, and obtain solutions for various usage cases. Refer to the **ADAPT-Builder User Manuals** and the **ADAPT-Builder 20 Tutorial** for additional information regarding Usage Cases and Stiffness Modifiers.

2.3.8 Selected EC2 Design Code Provisions Related to P-Delta Analysis

Eurocode 2 Section 5.8.2 states that global 2nd-order effects may be ignored if they are less than 10% of the first order effects. As an alternative, if the slenderness (l) is less than the slenderness limit (l_{lim}), then second order effects may be ignored. The main provision for this is as follows:

> (6) Second order effects may be ignored if they are less than 10 % of the corresponding first order effects. Simplified criteria are given for isolated members in 5.8.3.1 and for structures in 5.8.3.3.

Essentially, this is a definition of a threshold between a non-sway and sway structure.

The above limitation frees an engineer from more detailed slenderness design. The most adequate verification of this criterion can be obtained as a result of 2nd-order P-Delta analysis.

In case the 10% limits for standard analysis are exceeded and second order analysis is needed, Eurocode 2 specifies the following general requirements:

> (2)P Where second order effects are taken into account, see (6), equilibrium and resistance shall be verified in the deformed state. Deformations shall be calculated taking into account the relevant effects of cracking, non-linear material properties and creep.
>
> **Note**. In an analysis assuming linear material properties, this can be taken into account by means of reduced stiffness values, see 5.8.7.

This practically stipulates, that when analysis is based on assumption of linear materials and geometric nonlinearity, the reduced stiffness approach needs to be taken, similarly to the requirements of ACI code. Even though the detailed requirements may be different, the general ideas are similar in nature. From the standpoint of implementation in ADAPT-Builder's frame solver, the most interesting aspect is employing the use of usage cases with defined stiffness modifiers following code recommendations.

Eurocode 2 presents the provisions for 2nd-order analysis of individual isolated members and for entire structures (global analysis). With regards to the calculations performed by ADAPT-Builder's frame solver, provisions for isolated members are of no particular interest. The provisions for global analysis are of importance here.

Eurocode 2 mentions the following methods of analysis of slenderness effects:

5.8.5 Methods of analysis

(1) The methods of analysis include a general method, based on non-linear second order analysis, see 5.8.6 and the following two simplified methods:

(a) Method based on nominal stiffness, see 5.8.7
(b) Method based on nominal curvature, see 5.8.8

The general method is based on non-linear analysis, including geometric non-linearity i.e. second order effects. The provisions for general method (5.8.6) stipulate that both material and geometrical nonlinearity should be used. In case software does not handle material nonlinearity, these effects need to be accounted for by means of stiffness reductions, as was mentioned before, and should be based on requirements nominal stiffness method of section 5.8.7. The nominal stiffness method is primarily meant for global analysis. The nominal curvature method (5.8.8) is primarily to be used for isolated members, and therefore it is not useful in application within ADAPT-Builder.

The nominal stiffness method (5.8.7) presents the following requirements regarding calculations of stiffness reductions of slender elements:

(1) The following model may be used to estimate the nominal stiffness of slender compression members with arbitrary cross section:

$$EI = K_c E_{cd} I_c + K_s E_s I_s \qquad (5.21)$$

where:

E_{cd} is the design value of the modulus of elasticity of concrete, see 5.8.6 (3)
I_c is the moment of inertia of concrete cross section
E_s is the design value of the modulus of elasticity of reinforcement, 5.8.6 (3)
I_s is the second moment of area of reinforcement, about the centre of area of the concrete
K_c is a factor for effects of cracking, creep etc, see 5.8.7.2 (2) or (3)
K_s is a factor for contribution of reinforcement, see 5.8.7.2 (2) or (3)

(2) The following factors may be used in Expression (5.21), provided $\rho \geq 0{,}002$:

$$K_s = 1$$
$$K_c = k_1 k_2 / (1 + \varphi_{ef}) \quad (5.22)$$

where:

ρ is the geometric reinforcement ratio, A_s/A_c
A_s is the total area of reinforcement
A_c is the area of concrete section
φ_{ef} is the effective creep ratio, see 5.8.4
k_1 is a factor which depends on concrete strength class, Expression (5.23)
k_2 is a factor which depends on axial force and slenderness, Expression (5.24)

$$k_1 = \sqrt{f_{ck} / 20} \text{ (MPa)} \quad (5.23)$$

$$k_2 = n \cdot \frac{\lambda}{170} \leq 0{,}20 \quad (5.24)$$

where:

n is the relative axial force, $N_{Ed} / (A_c f_{cd})$
λ is the slenderness ratio, see 5.8.3

If the slenderness ratio λ is not defined, k_2 may be taken as

$$k_2 = n \cdot 0{,}30 \leq 0{,}20 \quad (5.25)$$

(3) As a simplified alternative, provided $\rho \geq 0{,}01$, the following factors may be used in Expression (5.21):

$$K_s = 0$$
$$K_c = 0{,}3 / (1 + 0{,}5\varphi_{ef}) \quad (5.26)$$

Note. The simplified alternative may be suitable as a preliminary step, followed by a more accurate calculation according to (2).

(4) In statically indeterminate structures, unfavourable effects of cracking in adjacent members should be taken into account. Expressions (5.21-5.26) are not generally applicable to such members. Partial cracking and tension stiffening may be taken into account e.g. according to 7.4.3. However, as a simplification, fully cracked sections may be assumed. The stiffness should be based on an effective concrete modulus:

$$E_{cd,eff} = E_{cd}/(1+\varphi_{ef}) \quad (5.27)$$

where:

E_{cd} is the design value of the modulus of elasticity according to 5.8.6 (3)
φ_{ef} is the effective creep ratio; same value as for columns may be used

The provisions of 5.8.7(4) are the most relevant from the standpoint of the implementation of P-Delta in ADAPT-Builder. This means that the stiffness reductions prescribed in eq. 5.27 (which are constant factors) are sufficient to satisfy the requirements of the EC2 code, for typical multi-story frame buildings. The value of effective creep ratio ϕ_{ef} should be determined based on requirements of sections 3.1.4 and 5.8.4.

Additional requirements regarding global 2nd-order analysis are presented in Annex H of EC2 code. In this annex, further simplifications are given as follows:

(3) In the absence of a more accurate evaluation of the stiffness, the following may be used for a bracing member with *cracked* section:

$$EI \approx 0,4\, E_{cd} I_c \quad \text{(H.3)}$$

where:
E_{cd} = E_{cm}/γ_{cE}, design value of concrete modulus, see 5.8.6 (3)
I_c second moment of area of bracing member

If the cross-section is shown to be *uncracked* in the ultimate limit state, constant 0,4 in Expression (H.3) may be replaced by 0,8.

Two stiffness reduction factors are mentioned here:

- $0.4*E_cI_c$ – for cracked elements
- $0.8*E_cI_c$ – for uncracked elements.

The use of these stiffness reductions is further elaborated in "Eurocode 2 Commentary" as follows:

The coefficient 0,4 (or 0,8) for estimating the stiffness (see H.1.2 (3)) can be compared to $0,3/(1+\varphi_{ef})$ in expression (5.26). Expression (5.26) is valid for *isolated members*, where all the vertical load considered acts on the member itself. Then there effects not only of cracking, but also of non-linearity in compression are considered. The last effect can be strong, particularly in cases where the section is uncracked, usually associated with high vertical load. For the same reason, a higher stiffness value for uncracked section is *not* given in 5.8.7.2. In a structure, on the other hand, most of the vertical load is normally on the *braced* units, which means that there is less effect of compression non-linearity on the *bracing* units, in which case a particular value for uncracked section (0,8) *is* justified[1]. A further difference is that the bending moment normally has a more favourable distribution in a bracing unit than in isolated members, which gives less overall effect of cracking. These circumstances together justify the use of 0,4/0,8 instead of $0,3/(1+\varphi ef)$. Creep is not included in the criterion for neglecting second order effects in structures (as it is for isolated members). The reason is that for global second order effects in structures, the dominating first order effect is wind. In this circumstance, there is little effect of creep, and consequently, the effective creep ratio according to 5.8.4 will be low.

In conclusion, the approach of reduced stiffness presented in EC2 is similar to the approach in ACI code, except for detailed values of reduction factors.

2.3.9 Selected CSA Design Code Provisions Related to P-Delta Analysis

The Canadian design code (CSA) is similar to ACI code with regards to 2^{nd}-order effects.

The values of stiffness reduction factors to be used in conjunction with P-Delta analysis are as follows:

10.14.1.2

The following properties may be used to determine the section properties specified in Clause 10.14.1.1:

Modulus of elasticity	E_c from Clause 8.6.2
Moment of inertia:	
Beams	$0.35I_g$
Columns	$0.70I_g$
Walls — uncracked	$0.70I_g$
Walls — cracked	$0.35I_g$
Flat plates and flat slabs	$0.25I_g$
Area	A_g

CSA code imposes the following limitation of displacement amplification factors (DAF):

10.16.5 Strength and stability checks

In addition to load cases involving lateral loads, the strength and stability of the structure as a whole under factored gravity loads shall be considered using the following criteria, with β_d based on the sustained axial load:

a) When $\delta_s M_s$ is computed as specified in Clause 10.16.3.1, the ratio of second-order lateral deflections to first-order lateral deflections under factored gravity loads plus a lateral load applied to each storey equal to 0.005 multiplied by the factored gravity load on that storey shall not exceed 2.5.

b) When δ_s is computed as specified in Clause 10.16.3.2, δ_s computed using $\sum P_f$ and $\sum P_c$ under factored gravity load shall be positive and shall not exceed 2.5.

These limitations are reported graphically and in XLS format within ADAPT-Builder's GUI for all design codes where P-Delta is used.

2.3.10 Selected NBR Design Code Provisions Related to P-Delta Analysis

Brazilian code defines non-sway and sway frames in a similar way to EC2, as follows:

15.4.2 Sway and non-sway structures

Non-sway structures are considered for design purposes as having fixed nodes when the horizontal displacements of the nodes are small and therefore the second order global effects are insignificant (lower than 10% of the respective first order internal forces). In these structures it is enough to consider the local and localized second order effects.

Sway structures are considered for design purposes as having mobile nodes where the horizontal displacements are not small and therefore the second order global effects are important (higher than 10% of the respective first order internal forces). In these structures the second order global internal forces and moments as well as those local and localized should be considered.

The following stiffness reduction factors are adopted for the purpose of elastic, 2nd-order analysis:

15.7.3 Approximate consideration of the physical non-linearity

To analyze second order global internal forces and moments in reticulated structures having at least four stories, the approximated physical non-linearity may be considered taking the rigidity of the structural elements with the following values:

— slabs : $(EI)_{sec} = 0.3E_{ci}I_c$

— beams: $(EI)_{sec} = 0.4E_{ci}I_c$ for $A_s' \neq A_s$ e

$(EI)_{sec} = 0.5\ E_{ci}I_c$ for $A_s' = A_s$

— columns: $(EI)_{sec} = 0.8E_{ci}I_c$

where:

I_c is the moment of inertia of the gross section of concrete including, whenever necessary, collaborating flanges.

When the bracing structure is made up only of beams and columns and when γ_z is smaller than 1.3, it is allowed to calculate the rigidity of beams and columns by:

$(EI)_{sec} = 0{,}7\ E_{ci}I_c$

The rigidity values adopted in this sub-section are approximated and cannot be used to estimate second order local internal forces and moments even with a finer discretization of the modeling.

2.3.11 Selected AS-3600 Code Provisions Related to P-Delta Analysis

Australian code AS-3600 presents the following requirements regarding 2nd-order P-Delta analysis:

6.3.2 Analysis

An elastic analysis incorporating secondary bending moments shall comply with the requirements of Clause 6.2 and the following:

(a) The effect of lateral joint displacements shall be taken into account.

(b) For strength design of a regular rectangular framed structure, the cross-sectional stiffness of the flexural members and columns may be taken as $0.4E_cI_f$ and $0.8E_cI_c$ respectively.

(c) For very slender members, the change in bending stiffness of a member due to axial compression shall be considered.

Stiffness reduction factors for this code are similar to EC2.

2.3.12 Design Codes – P-Delta Summary

In summary, based on the above, when performing global 2nd-order analysis based on ACI or EC2, NBR, AS-3600, it is permissible to account for material nonlinearity (cracking, creep etc) using simplified stiffness reduction factors. The basic reduction factors for non-prestressed elements are as follows:

	ACI 318 & CSA	EC2, AS-3600	NBR
Beams	$0.35*E_cI_c$	$0.4*E_cI_c$	$0.4*E_cI_c$
Columns	$0.70*E_cI_c$	$0.8*E_cI_c$	$0.8*E_cI_c$
Walls - uncracked	$0.70*E_cI_c$	$0.8*E_cI_c$	$0.8*E_cI_c$
Walls - cracked	$0.35*E_cI_c$	$0.4*E_cI_c$	$0.4*E_cI_c$
Flat Plates and Slabs - cracked	$0.25*E_cI_c$	$0.4*E_cI_c$	$0.3*E_cI_c$

ADAPT-Builder gives the user the capability of defining customized, user-imposed “Usage Cases” where stiffness modifiers are introduced and imposed at a component level.

While some design codes don’t impose an explicit limitation on the amplification factors obtained in the course of P-Delta analysis (stability limit), ADAPT-Builder reports a ratio of $2^{nd}/1^{st}$ order drifts and moments. The default limit in ADAPT-Builder is set to 1.4 but can be modified by the user. The program checks this and graphically reports the outcome. XLS data is also presented for drift results.

2.4 Drift and Moment Amplification Factors

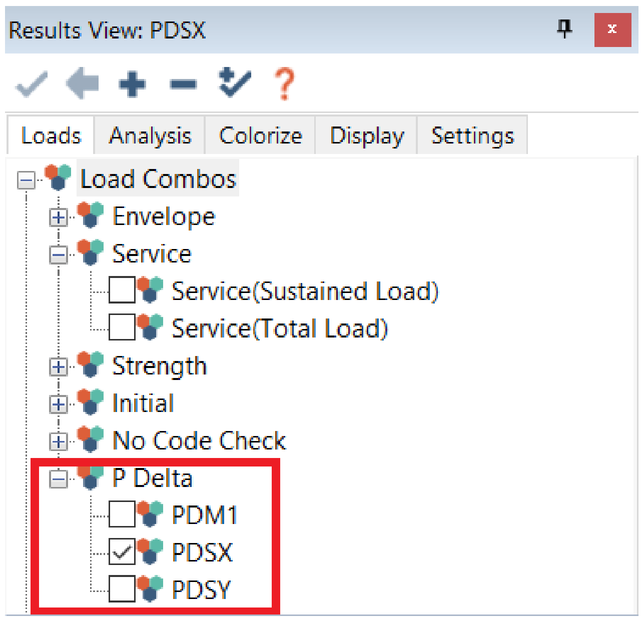
2.4.1 Introduction

With the new P-Delta feature, the program graphically reports drift and moment amplification factors (2nd/1st order results) for local axes (RR and SS) as well as the combined global results. The program code checks against a user-defined value set in the Result Browser. The default value is described in the section above.

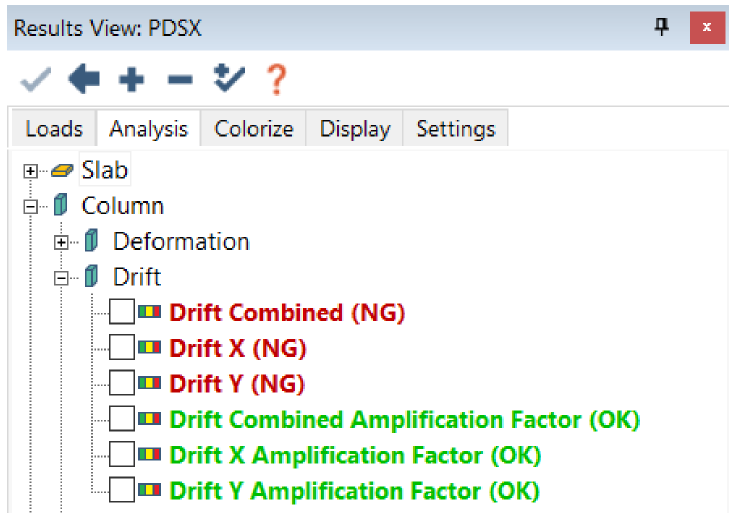
Some design codes require an explicit limitation on drift amplification for the purpose of ensuring a stability limit. The following sections describe how the drift and moment amplification factors (DAF and MAF) are used. The prerequisite to these results is to have completed a global FEM solution with the presence of P-Delta combinations included.

2.4.2 Graphical Drift Amplification

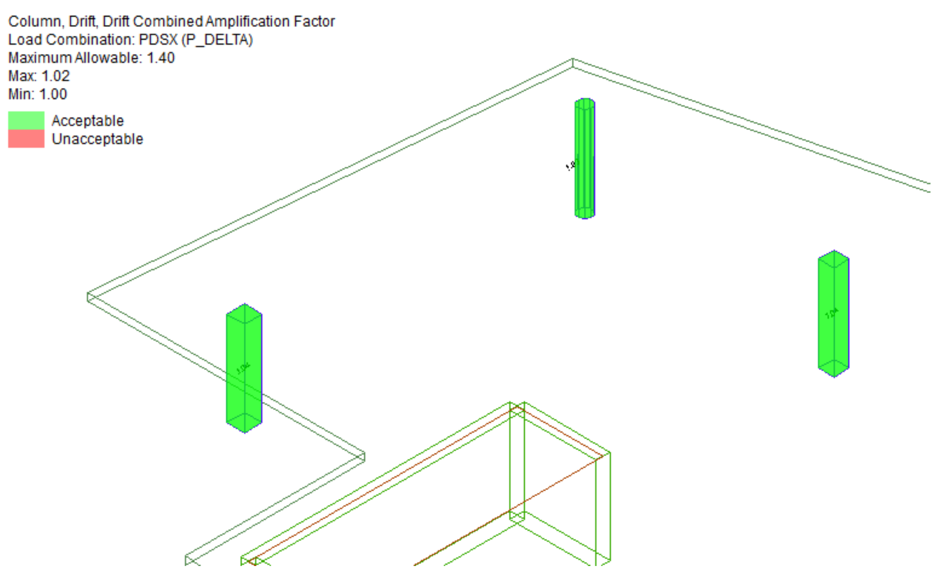
To invoke the graphical drift amplification values and code check, the user must select a P-Delta combination result in the 'Results Browser' as shown below. Note that selection of the master P-Delta combination is usually not required or typical for drift amplification since it serves only as the driver for the geometric stiffness generation.



Once the combination has been selected, the 'Analysis' tab can be used to produce the result. Select Column→Drift and the amplification factor options are selectable for Combined, X, or Y directions. Once selected the program will produce a Green or Red colored column to indicate if the value is less than or greater/equal than the user-defined value. This value is set to 1.4 by default. The user can change this setting in the 'Display' tab from the Result Browser.



The image belows shows the code check for the combined drift amplificaton factors at the top level of a structure. Note the legend in the upper corner of the window reports the result, combination and type, allowable value and the max/min values.

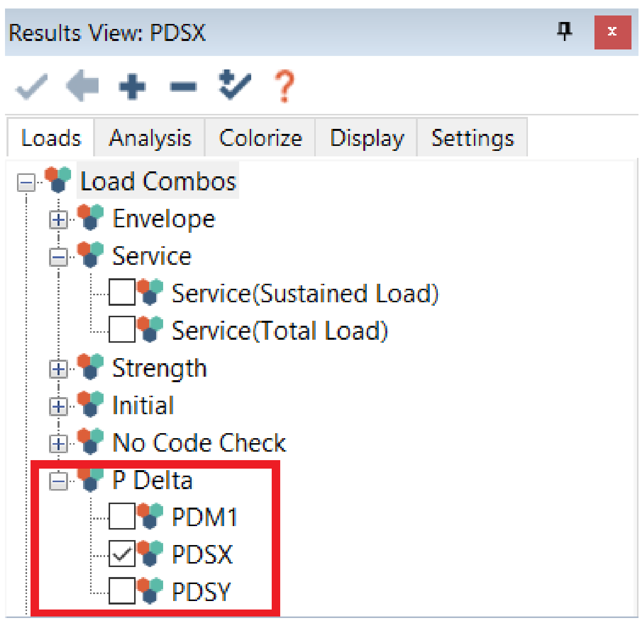


To produce this result, the program calculates the drift for both the 1st and 2nd order results. Drift is taken as the differential displacement from the top node of the upper frame element of the column to the bottom node of the bottom frame element of the column, divided by story height. The ratio is then taken between the calculated drifts for 2nd to 1st order results. Note that if a situation is modeled where multiple columns extend over multiple floors and don't connect to slabs at each column break-point, the drift is still reported per column and the height used is that between the top and bottom of column.

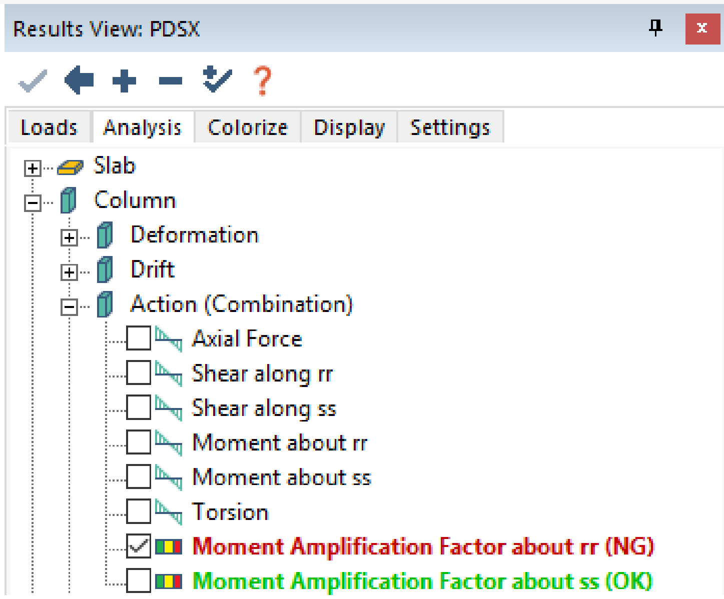
Displacements used for the drift calculations are those relative to the result selection made: X, Y or combined. For combined the program uses the global displacements.

2.4.3 Graphical Moment Amplification

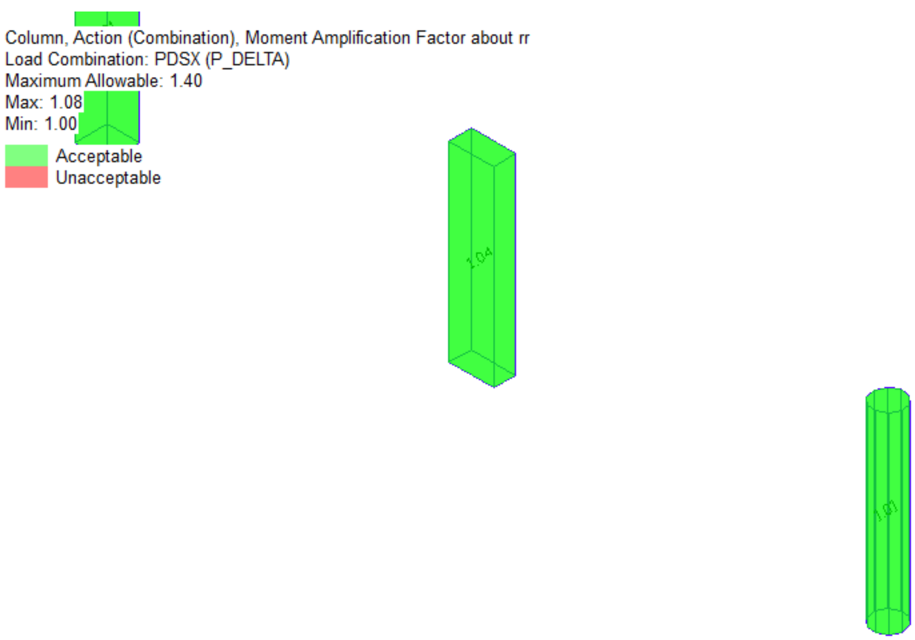
To invoke the graphical moment amplification values and code check, the user must select a P-Delta combination result in the 'Results Browser' as shown below. Note that selection of the master P-Delta combination is usually not required or typical for moment amplification since it serves only as the driver for the geometric stiffness generation.



Once the combination has been selected, the 'Analysis' tab can be used to produce the result. Select Column→Action (Combination)→Moment Amplification Factor about rr or ss. The amplification factor options are selectable for moments about rr or ss local axes for columns. Once selected the program will produce a Green or Red colored column to indicate if the value is less than or greater/equal than the user-defined value. This value is set to 1.4. The user can change this setting in the 'Display' tab from the Result Browser.



The image belows shows the code check for the moment amplificaton factor about the local rr axss at the top level of a structure. Note the legend in the upper corner of the window reports the result, combination and type, allowable value and the max/min values.



To produce this result, the program calculates the moments at the top and bottom of columns for both the 1st and 2nd order results. The ratio is then calculated for the top and bottom locations for 2nd to 1st order results. The maximum ratio is reported and checked against allowable.

2.5 Effective Flange for Beam Design in ADAPT-Builder

2.5.1 Introduction

The program now includes the option to calculate the effective flange and properties of beams when the support line is set to 'Beam' criteria. The design of the beam sections is performed using the effective properties. The component of stress related to the precompression uses the full tributary area of the section as does the graphical precompression result. ACI and EC2 calculations for effective width are supported.

2.5.2 Methodology

The program includes effective flange formulation for two design codes: ACI318-14 and EC2. All design codes will default to the ACI methodology expect for EC2 and NBR codes. The user can also manually generate splitters or use max tributary offset input for the design strip to create user-defined flange widths.

The implementation for ACI uses the following expressions from ACI318-14 Section 6.3.2, excluding point (c).

Effective width of the flange[36]

i. For T-Beams

Effective overhanging flange width on each side of web is the smallest of:

a. Ln/8;
b. 8 times the flange thickness;
c. ½ of the clear distance to the next web – not checked in program

ii. For L-Beams

Effective overhanging flange width on each side is the smallest of:

a. Ln/12;
b. 6 times the flange thickness;
c. ½ of the clear distance to the next web – not checked in program

The implementation for EC2 uses the following expressions from EN 1992-1-1:2004 Section 5.3.2.1 (3).

(3) The effective flange width b_{eff} for a T beam or L beam may be derived as:

$$b_{eff} = \sum b_{eff,i} + b_w \leq b \qquad (5.7)$$

where

$$b_{eff,i} = 0,2b_i + 0,1l_0 \leq 0,2l_0 \qquad (5.7a)$$

and

$$b_{eff,i} \leq b_i \qquad (5.7b)$$

(for the notations see Figures 5.2 above and 5.3 below).

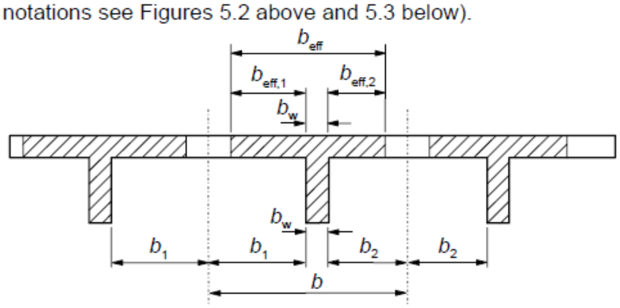


Figure 5.3: Effective flange width parameters

2.5.3 Defining the Effective Flange Option in a Model

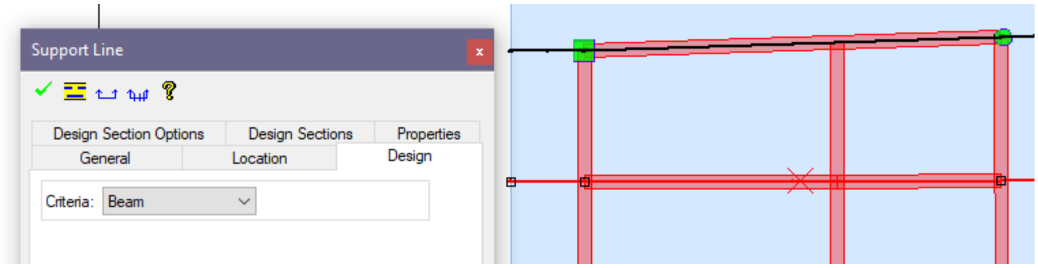
When the effective flange option is used, the program uses effective properties in performing the following calculations:

- Calculation of reinforcement for minimum and strength requirements of the design section
- Calculation of flexural stresses for PT design sections. That component of the stress calculation, *My/I*, is based on the centroid of the effective section and moment of inertia of the effective section. That component of the stress calculation, *P/A*, uses the area based on the full design section as modeled for the design strip. Note that the P/A as reported graphically is based on the full tributary area, not the effective area.

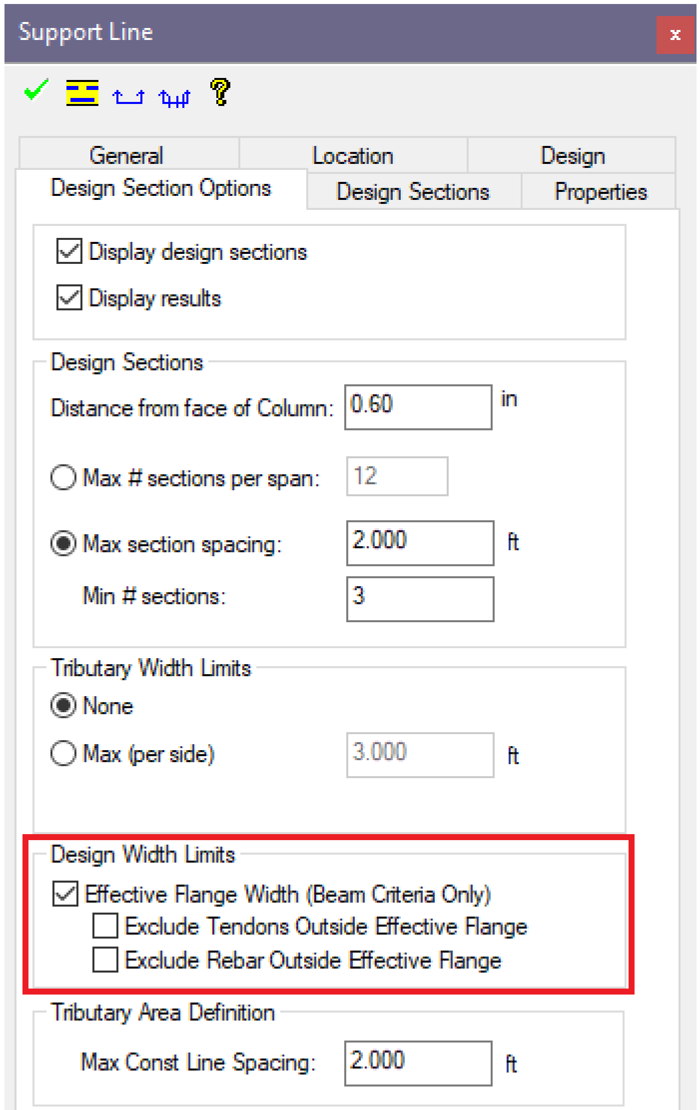
Actions for the design of effective sections are those as calculated per the Nodal Integration Method (see ADAPT-Technical Note TN302 – Design Section Values) over the full tributary width of the design section.

To define use of the effective flange in the ADAPT model, use the following sequence of instructions:

1. Set the support line to the 'Beam' design type from double-clicking on the support line, selecting 'Design' and set the criteria to 'Beam.'

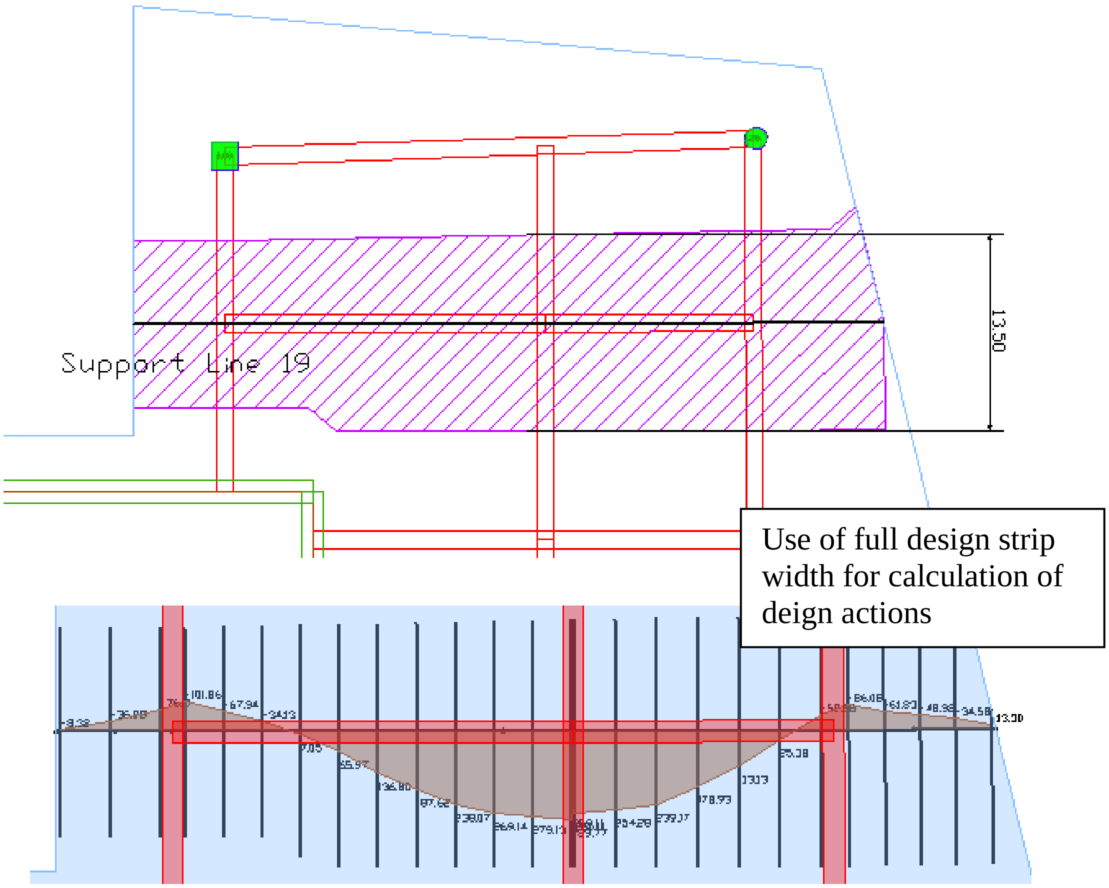


2. Set the ‘Design Section Options’ for the support line. Use the options in ‘Design Width Limits’ to enable the option for effective flange. Two additional options are shown for exlusion of tendons and rebar outside of the effective flange. If these options are selected, the program will ignore the area of prestressing or reinforcement that fall within the the modeled tributary but outside effective flange, for calculation of reinforcement for the design section.



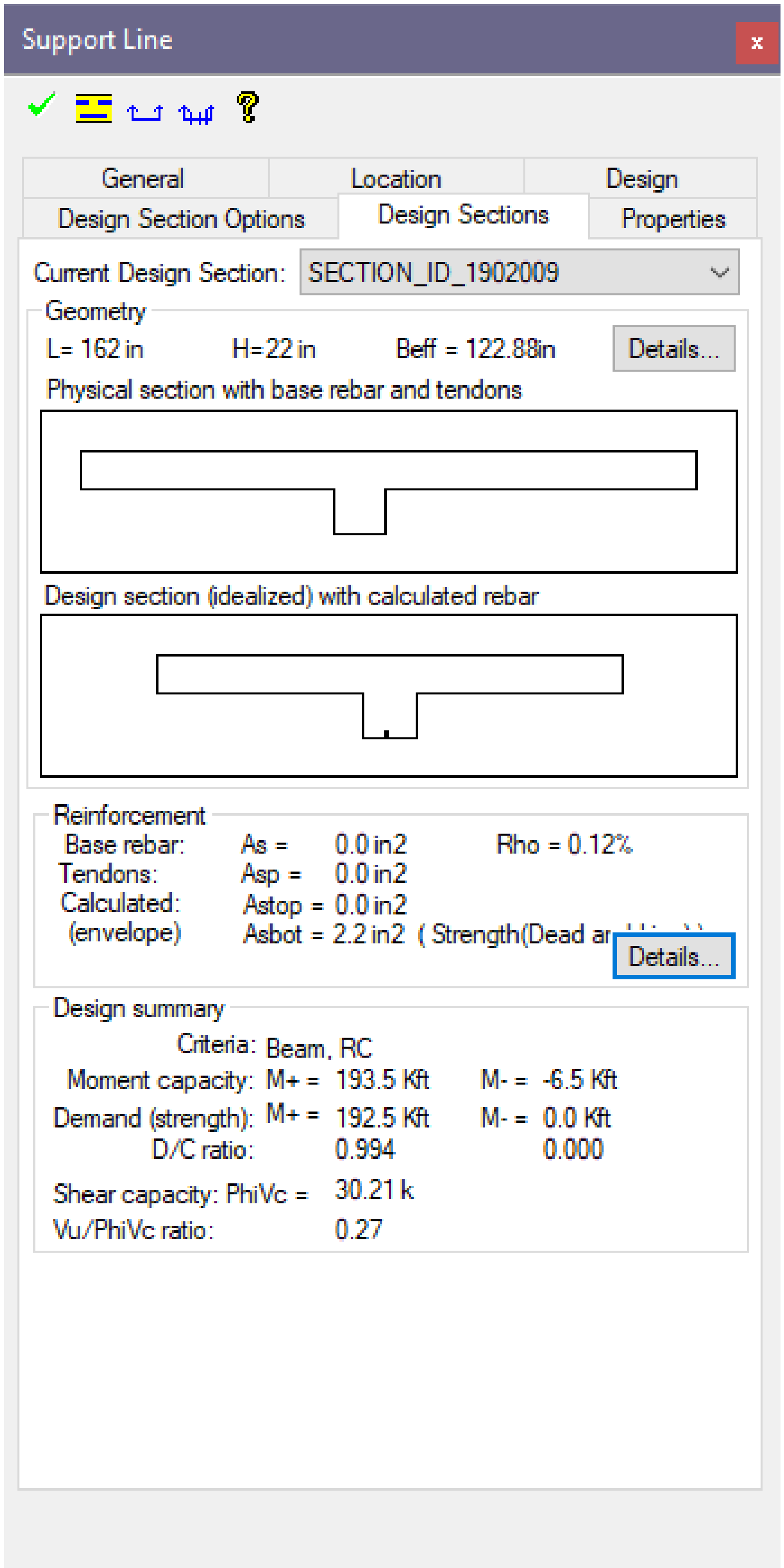
3. Select the green checkbox at the top left corner of the Support Line property box.

4. Generate the design sections from Floor Design→Generate Design Sections. Note for the image below the generated design strip is taken as the full tributary of 13.5 ft (162 in). The design actions at the centroid of the design section is based on nodal integration of the actions over the full width of the generated design strip.



The image below shows the properties of a design section with consideration of effective flange properties. While the generated design strip width is 162in., the effective flange width as used in the calculation of reinforcement and flexural stress is taken and calculated as 122.9in.

The window below shows two images. The first is the physical section cut and the second is the idealized section cut which uses the effective flange.


Support Line
General
Location
Design
Design Section Options
Design Sections
Properties
Current Design Section: SECTION_ID_1902009
Geometry
L= 162 in
H=22 in
Beff = 122.88in
Details...
Physical section with base rebar and tendons
Design section (idealized) with calculated rebar
Reinforcement
Base rebar: As = 0.0 in2
Rho = 0.12%
Tendons: Asp = 0.0 in2
Calculated: Astop = 0.0 in2
(envelope) Asbot = 2.2 in2 (Strength(Dead ar
Details...
Design summary
Criteria: Beam, RC
Moment capacity: M+ = 193.5 Kft M- = -6.5 Kft
Demand (strength): M+ = 192.5 Kft M- = 0.0 Kft
D/C ratio: 0.994 0.000
Shear capacity: PhiVc = 30.21 k
Vu/PhiVc ratio: 0.27

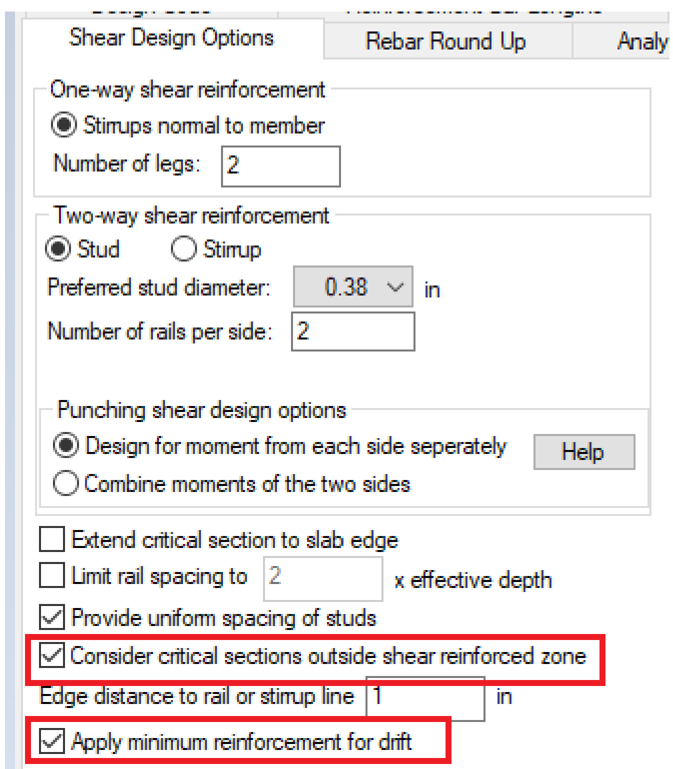
2.6 Punching shear for Aci 318-14

2.6.1 Introduction

New improvements have been made for punching shear checks in ADAPT-Builder according to ACI318-14. The first improvement is the option to consider the least critical section as an octagonally-shaped section at a distance of d/2 outside the last line of reinforcement. The applicable code sections being 22.6.4.2 and 22.6.6.1.

The second improvement is the option to consider minimum reinforcement up to a distance 4*slab thickness to satisfy a minimum strength of 3.5*f'c^1/2. This requirement is found is Section 18.14.5.1 and is applied to satisfy adequate ductility of members not part of a lateral-resisting system in seismic design categories D, E, and F.

The program provides the user the option to consider the preceding two features from Floor Design→Shear Options. The options can be considered in tandem (as shown below) or separate.



The third improvement applies to the application of minimum reinforcement at critical sections as dictated in ACI318-14 Section 22.6.8.3. The critical section used to determine the minimum amount of reinforcement is that of the section located d/2 from face of support. In previous versions of the software, the critical section length as based on the section being checked. In other words, the amount of reinforcement necessary to meet minimum became larger each critical section (layer) that was checked. Upon a formal review of the program source code and the interpretation of design code, this was modified and improved.

The following sections provide more detailed description of the first two features described above.

2.6.2 Critical Section Outside the Reinforced Zone

The option to consider critical sections outside the shear reinforced zone with either a rectilinear or octagonal-shaped critical section is given in the program. ACI318-14 Sections 22.6.4.2, 22.6.6.1 apply. The following images show the octagonal-shaped section introduced in ADAPT-Builder 2019 for compliance with ACI318-14. Note this section shape can now also be used for designs performed using all ACI318 versions supported by the program.

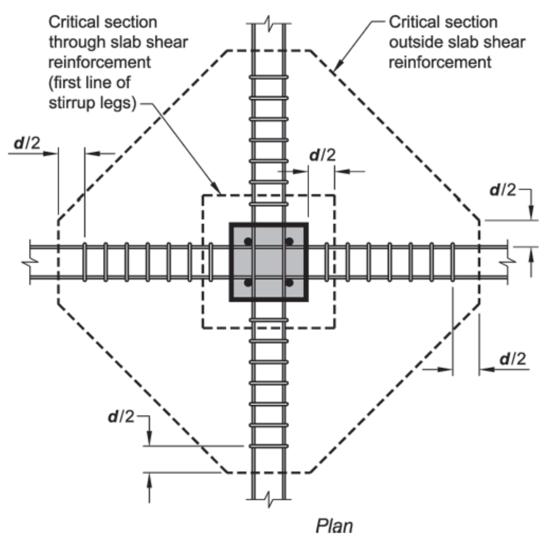


Fig. R22.6.4.2a—Critical sections for two-way shear in slab with shear reinforcement at interior column.

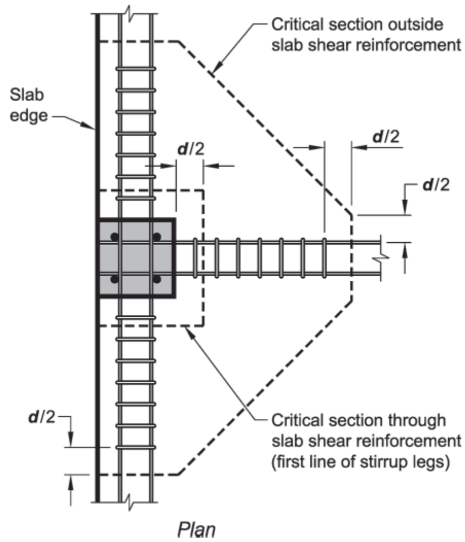


Fig. R22.6.4.2b—Critical sections for two-way shear in slab with shear reinforcement at edge column.

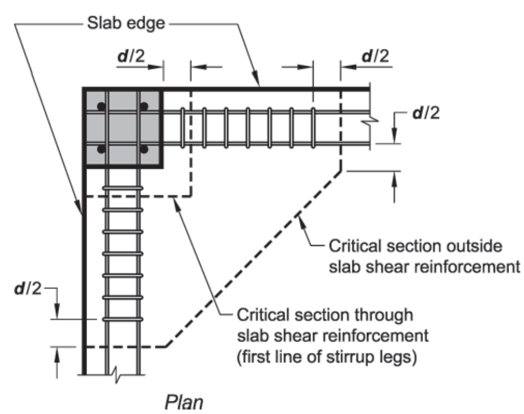
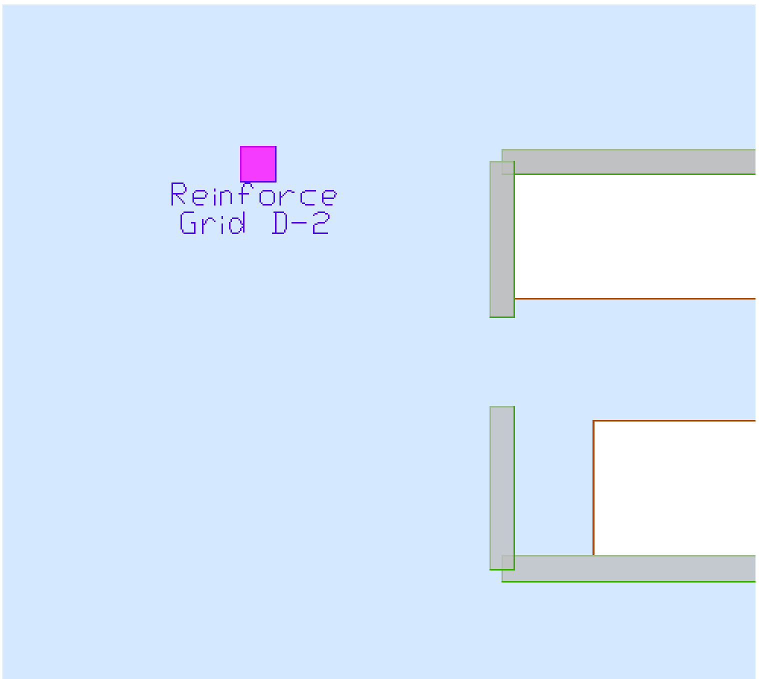


Fig. R22.6.4.2c—Critical sections for two-way shear in slab with shear reinforcement at corner column.

When this option is selected the program determines the critical section, based on a rectilinear shape, where no reinforcement is required. Each critical section is located at d/2 from the previous. At the critical section where no reinforcement is required (As = 0), the program rechecks the same location but with a reduced critical length and area. This is calculated using an octagonally-shaped section and is checked until the stress is less than code specified. The user can specify the edge distance between the edge of column and stud rail or shear stirrups. This value affects the critical section length.

The location where at which the octagonally-shaped critical section begins being checks is evident in the XLS output for punching shear in ADAPT-Builder. This can be viewed from Reports→Punching Shear→XLS Reports→Punching Shear Report. There are two sheets within this report that dictate the location the of the section shape change. These include the 'Critical Section Geometry' and the 'As Required' sheets. An example is presented below for an interior column located at Grid D-2. The first image is a graphical indication that this column requires reinforcement. Continuing into the details of the XLS report, the 2nd image below shows the location at which the critical section area reduces. Note between critical sections 1 and 5 the area increases. Between sections 5 and 6 there is a decrease in area from 1424.8in2 to 1226.9in2. This indicates that section 6 is the first section checked as an octagonal section. Additional sections (8-14) are required to be checked with the same shape.



Grid D-2	56	Interior	1	FOS	774.53
			2	FOS	937.09
			3	FOS	1099.67
			4	FOS	1262.25
			5	FOS	1424.81
			6	FOS	1226.88
			7	FOS	1341.84
			8	FOS	1456.78
			9	FOS	1571.66
			10	FOS	1686.67
			11	FOS	1801.68
			12	FOS	1916.53
			13	FOS	2031.54
			14	FOS	2146.55
Grid B-1	57	Interior	1	FOS	723.54

The image below shows the required area of steel for the same column at Grid D-2. Note that at critical section 5, where the rectilinear shape is used, the required As is calculated as zero. The next critical section and all remaining in sequence are those checked for the octagonally-shaped section.

The user can also review the 'Stress Check' sheet to review the allowable stress reported and there is clear indication given where the maximum stress of ϕ*2*f'c$^{\wedge 1/2}$ is reported. See table 22.6.6.1 for reference.

Grid D-2	56	Envelope	14	1.580965	1.728211	2.027355	2.328048	2.328048	2.262949	2.473745	2.68609	2.
		Strength(D	14	1.580965	1.728211	2.027355	2.328048	2.328048	0	2.473745	2.68609	2.
		Strength(D	9	1.428138	1.428138	0	1.838259	2.050604	2.262949	2.473745	2.68609	

2.6.3 Minimum Reinforcement for Drift

The option to apply two-way shear provisions for minimum reinforcement for seismic drift as found in ACI318-14 Section 18.14.5.1 is now included in ADAPT-Builder. The referenced provision is found in the excerpt below. Note that the provision is conditional upon a check of the interstory drift ratio at the support location. However, the program does not check this but gives the user the option to directly enforce the minimum shear strength as given as 3.5*f'c$^{\wedge 1/2}$.

18.14.5 *Slab-column connections*

18.14.5.1 For slab-column connections of two-way slabs without beams, slab shear reinforcement satisfying the requirements of 8.7.6 or 8.7.7 shall be provided at any slab critical section defined in 22.6.4.1 if $\Delta_x/h_{sx} \geq$ **0.035 – (1/20)** $(v_{ug}/\phi v_c)$. Required slab shear reinforcement shall provide $v_s \geq 3.5\sqrt{f_c'}$ at the slab critical section and shall extend at least four times the slab thickness from the face of the support adjacent to the slab critical section. The shear reinforcement requirements of this provision shall not apply if $\Delta_x/h_{sx} \leq$ **0.005**.

The value of (Δ_x/h_{sx}) shall be taken as the greater of the values of the adjacent stories above and below the slab-column connection. v_c shall be calculated in accordance with 22.6.5. v_{ug} is the factored shear stress on the slab critical section for two-way action due to gravity loads without moment transfer.

The current expresssions used to calculate As for two-way shear is shown below as taken from ADAPT-Technical Note TN504. If the new minimum reinforcement for drift requirement is selected, the program checks that the quantity in the parentheses is ≥ 3.5*f'c$^{1/2}$ for all critical sections with a distance 4*slab thickness from face of the support. Additional checks with respect to Av,min still apply as shown below.

$$A_v = \frac{(v_u - \varphi_v v_c)\ us}{\varphi_v f_y \sin(\alpha)}$$

For studs, $A_v \geq A_{vmin}$[24] where $A_{vmin} = \frac{2\sqrt{f'_c}us}{f_y}$

Where,

v_c = 2*λ*√f'c[25] [0.17*λ*√f'c in SI] for stirrups

= 3*λ*√f'c[26] [0.25*λ*√f'c in Si] for studs

α is the angle of shear reinforcement with the plane of slab and u is the periphery of the critical section. s is the spacing between the critical sections [d/2].

If required, shear reinforcement will be extended to the section where v_u is not greater than Φ_v*2λ*√f'c [Φ_v*0.17λ*√f'c in SI].

2.7 FELT 3D and Long-Term Losses

2.7.1 Introduction

When the PT Shop Drawing module is enabled in the splash screen of ADAPT-Builder 2019, the program enables new input and functionality for the calculation of long-term losses for unbonded tendons and bonded tendons stressed at the same time.

Explicit calculation of long-term losses related to elastic shortening, creep, relaxation, and shrinkage can now be performed for all modeled tendons in ADAPT-Builder. This can be performed in addition to friction loss and elongation calculations. The program retains the ability to enter lump-sum long-term loss estimates as an alternate method. Detailed loss calculation examples and background can be found in ADAPT-Technical Note TN-T9-04-Prestresssing Losses and Elongation Calculations.

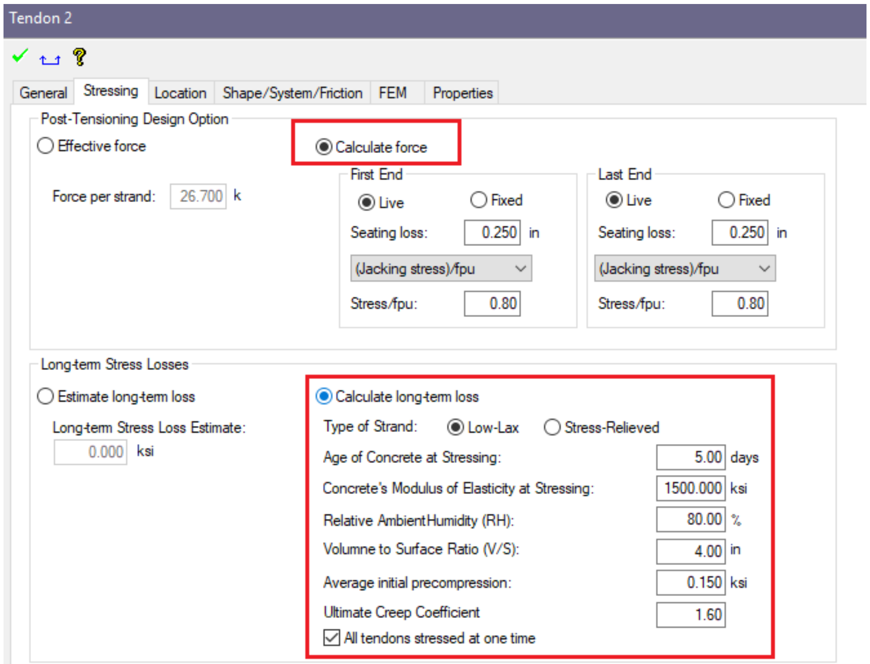
The program provides detailed tabular and graphical report data for FELT 3D. The reporting options are located in a later section of this document.

Additional features have been added to a new Drawing Editor. This option is also invoked when the user enables the PT Shop/3D FELT module in the program splash screen. The option can be found from Tendons→Shop Drawing→Drawing Editor. The information below provides more detail about each of these features including a new tendon shape for modeling and representation of tendons with in-plane sweep.

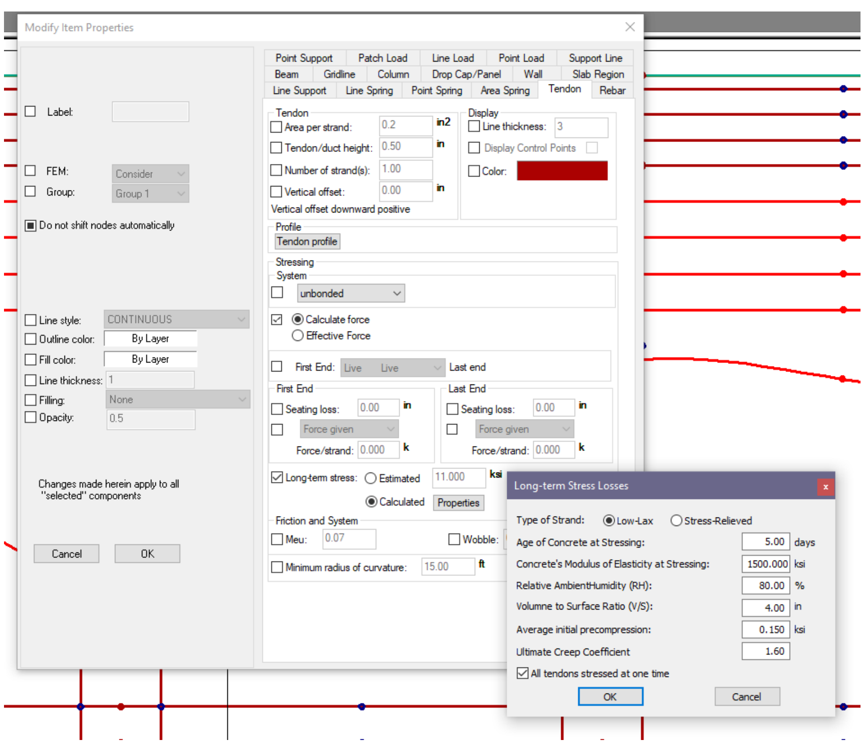
2.7.2 Setting Options for Long-Term Loss Calculations

The option to calculate long-term losses for modeled tendons can be set individually or by selecting a group of tendons. Remember that unless the option for the PT Shop/3D Felt is enabled, long-term losses can only be entered as lump sum values.

Individual Tendon: Double-click on a tendon to open the Tendon Properties dialog. In the ‘Stressing’ tab the option checked for ‘Calculate force’ will enable the long-term loss options. Selecting ‘Calculate long-term loss’ will enable the user to input the long-term loss parameters as shown in the image below.

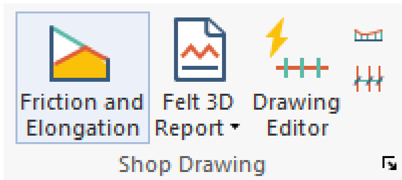


Multiple Tendons: Select a group of tendons and go to Modify→Modify Item Properties→Tendon. In the 'Stressing' tab select Calculated→Long-Term Stress→Calculated→Properties. This will enable the user to input the long-term loss parameters as shown in the image below and apply these to the selected group of tendons.

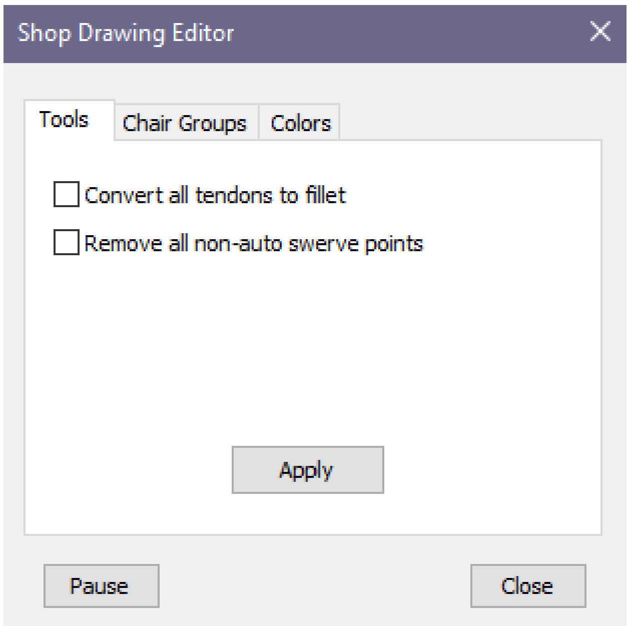


2.7.3 Drawing Editor

To invoke the Drawing Editor, select the icon shown below from Tendon→Shop Drawing.



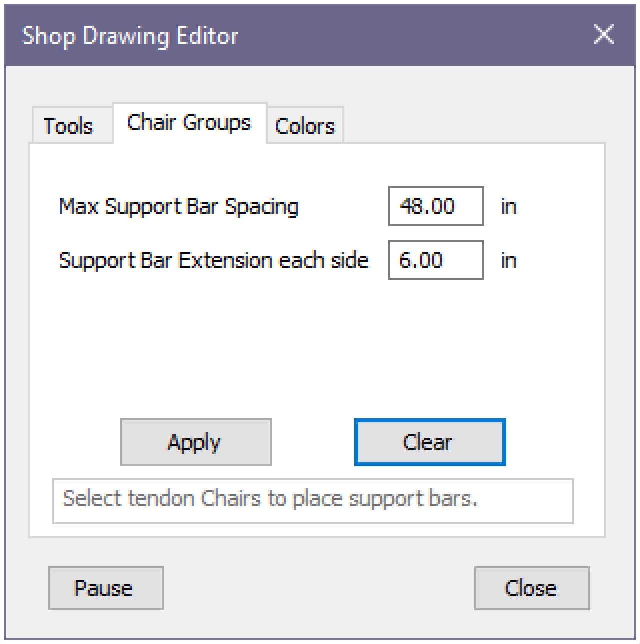
The Shop Drawing Editor contains 3 primary tabs as shown below: Tools, Chair Groups, and Colors.

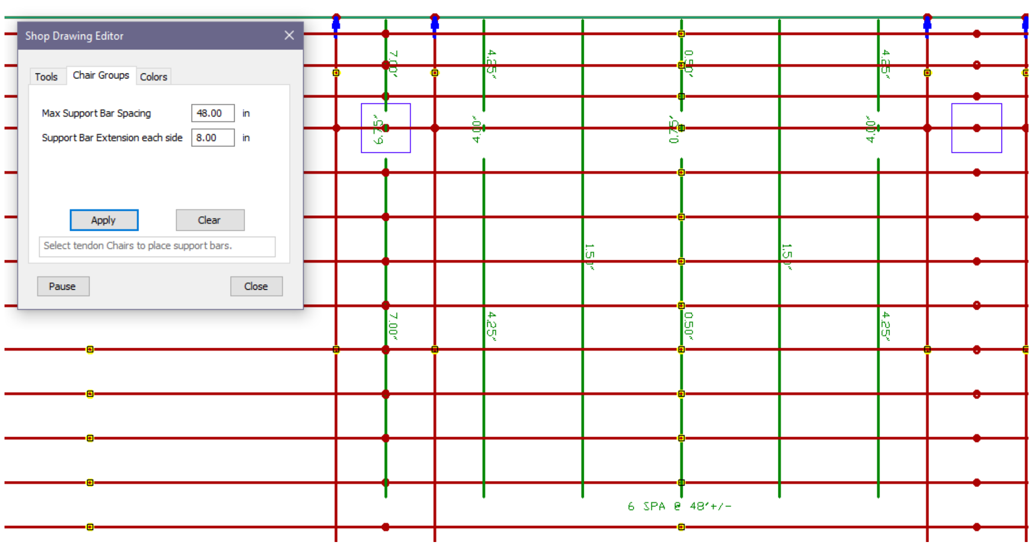


Tools – This option allows the user to convert the all tendons from straight or spline representation to fillet-radius representation. This is described in section 2.5.4. The user can also remove non-auto swerve points for tendons set to spline mode using this option. By selecting 'Apply' the program will remove the points for all tendons set to spline meeting this condition.

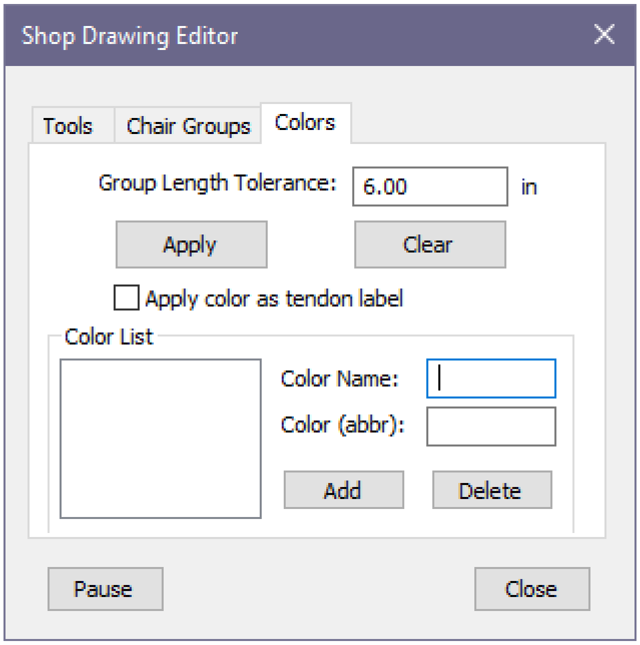
Chair Groups – This option allows the user to input on-demand chair bar maximum spacing and chair bar extensions for graphical output of tendon support bars and chair heights. To remove the bars and chair heights, use the 'Clear' option and select the desired tendons.

In the 2nd image below, max bar spacing is set to 48" with the bar extension each side set to 8 inches. The example shows distributed tendons. Where the support bars overlap, the bars are represented as continuous with the applicable chair height.





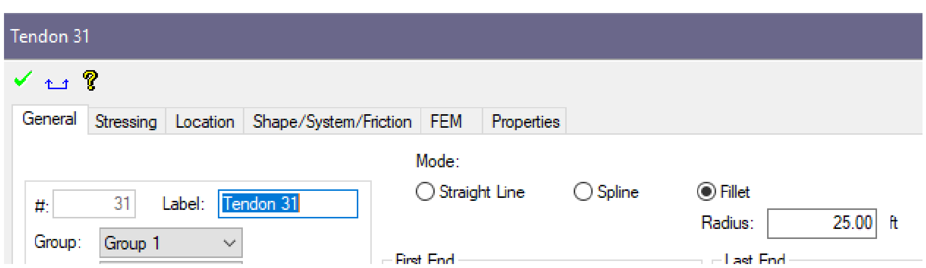
Colors – This option provides input for grouping tendons into user-defined colors and defining tendon labels per the color group assigned to the tendon. The program combines tendons of the same installation length into groups for based on the user-defined colors. New graphical tendon properties related to colors, groups, labels and lengths can be found from Tendon→Display Manager.



2.7.4 Tendon Shape – Fillet-Radius

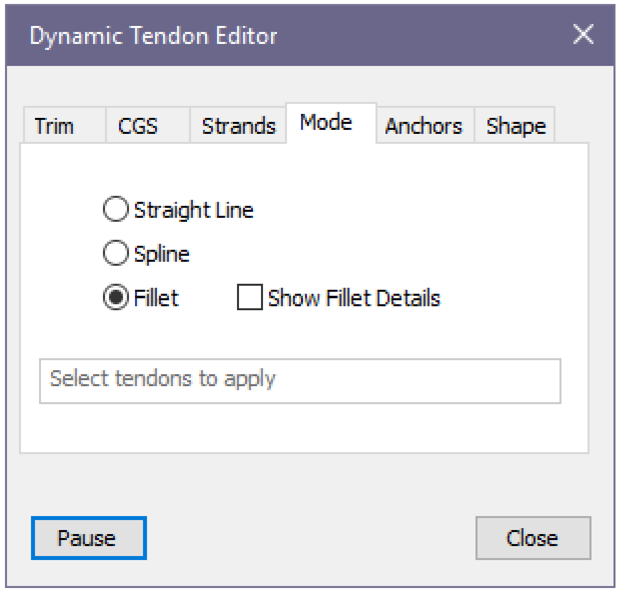
The program now includes the option to geometrically and analytically represent the tendon as ‘Fillet-Radius’ input. This combines straight tendon segments with curves represented as fillets based on user-input for the radius. The option allows the user to control in-plane deviation at control points without auto-correction of adjacent spans similar to how spline tendons are treated.

Individual Tendon: Double-click on a tendon to open the Tendon Properties dialog. In the ‘General’ select the ‘Fillet’ option and input the radius from the ‘Mode’ selections. The radius is measured from the an origin of the control point that has been deviated in-plane.



Multiple Tendons: Select a group of tendons and go to Tendon→Tendon Editor. This will open the Dynamic Tendon

Editor. In this dialog window select the 'Mode' tab to change to the Fillet option. This will apply this mode to all selected tendons.



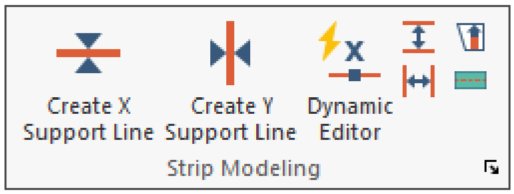
Chapter 3

MODELING IMPROVEMENTS

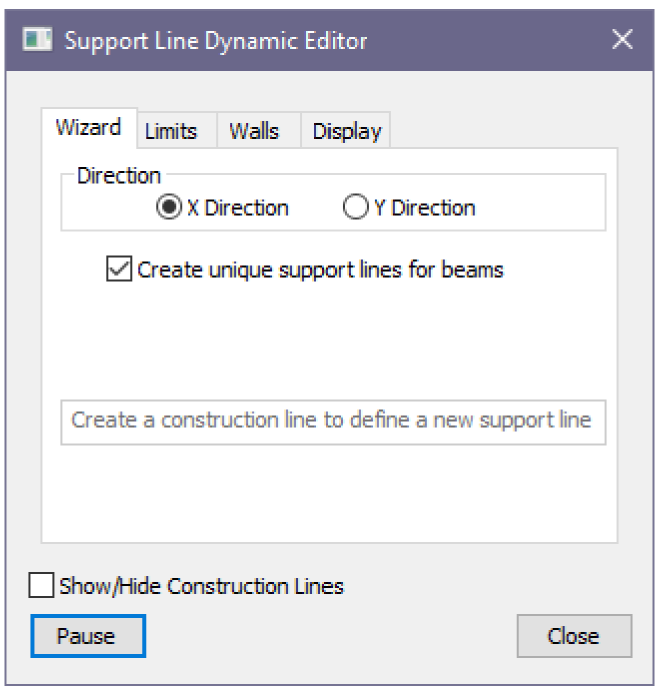
3. MODELING IMPROVEMENTS

3.1 Strip Modeling – Dynamic Editor

A new and Strip Modeling Dynamic Editor includes tools used to more rapidly model and modify support lines and design strips. This new tool is found at Floor Design→Strip Modeling. Manual creation of support lines is still available in v2019 as it has been done in previous versions through use of the Create X and Create Y Support Line tools.



The image below shows the Dynamic Editor dialog window. Note the option is given to 'Pause' or 'Resume.' This allows the user to keep the window open and work in the main User Interface if needed.



Tools included are in the Dynamic Editor are described below.

- **Support Line Wizard** – Creates a support based on a construction line defined by snap points along the strip path. Rules included in placement of construction line vertices include:

    1. The first snap point defines the start point (vertices) of the support line.

    2. Select ENTER or right-click and select Close/End/Accept to complete the construction line.

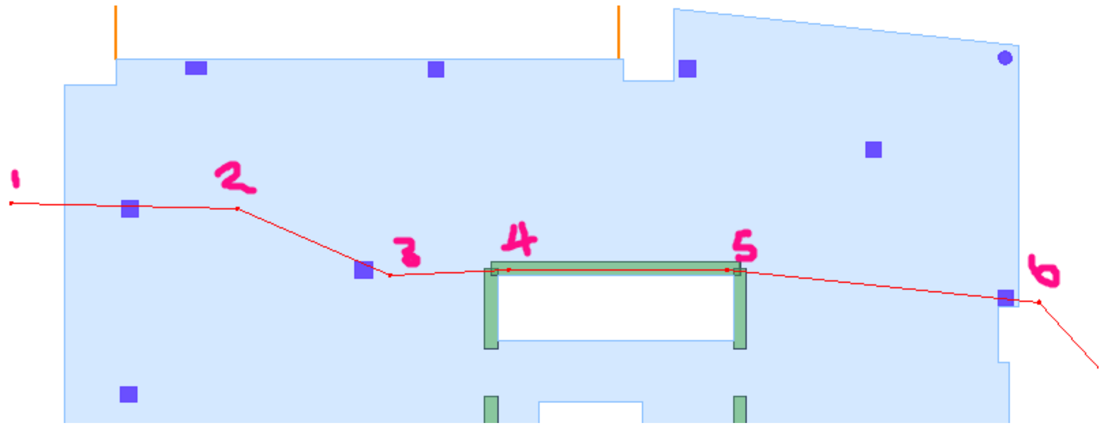
    3. Upon completion of a construction line, the program works in continue mode and the next snap dictates the start of the next construction line. This happens until the use exits the Dynamic Editor or moves to a different tool tab in the window.

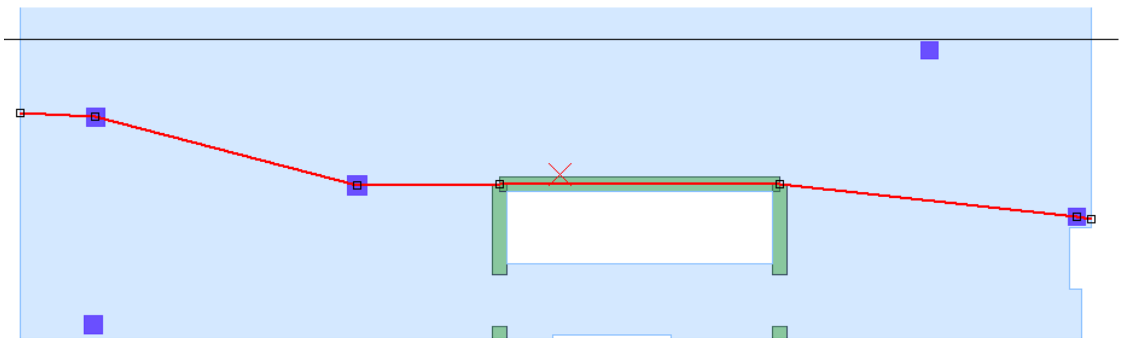
    4. Beginning and end construction line snap points that start or terminate outside of a slab edge will be auto-trimmed back to the slab edge.

    5. Snap points are not required to click on the centroid of a column for the program to create a vertex at that point. The construction line only be required to pass through a column.

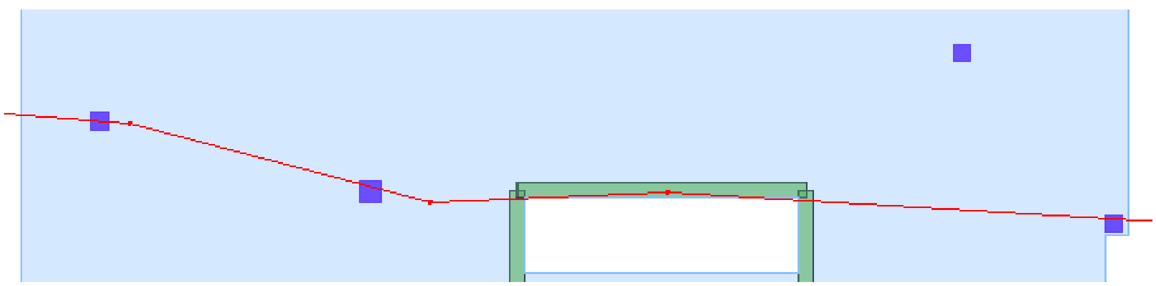
    6. The construction line can pass through multiple columns in a single segment of the construction line. A construction line vertex allows the construction line to change direction. In the image below the construction line begins at point 1 and passes through the first column. The segment between points 2 and 3 also pass through the 2nd column. In both cases, because the segments intersect the column, the program places support line vertex at these support locations. The 2nd image below shows the created support line.

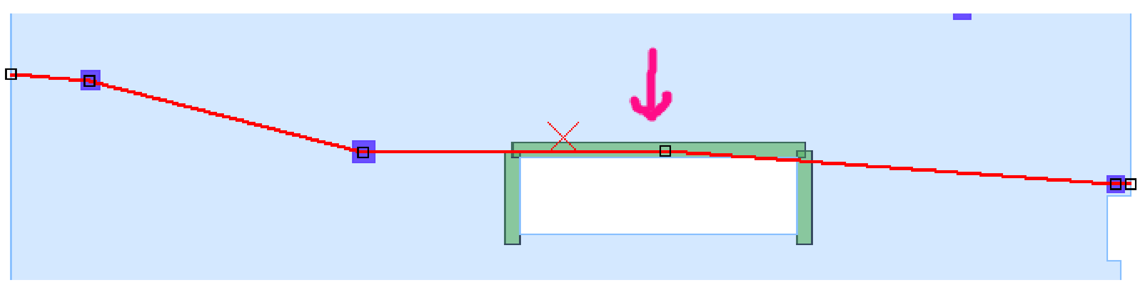
    7. The construction line vertices should snap near end-points of the wall for the auto-generation of support line nodes at the wall ends. The first image below shows the points placed near wall end and the outcome includes the nodes at the wall endpoints.





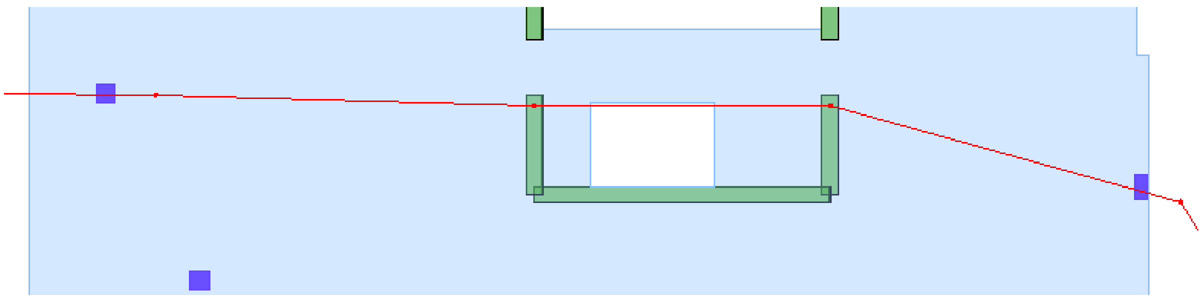
8. If the construction line vertex snaps in the middle half of a wall, only a single node is added at that snap point.

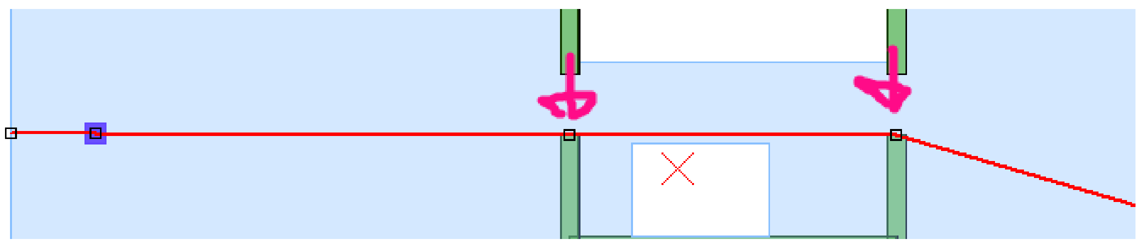




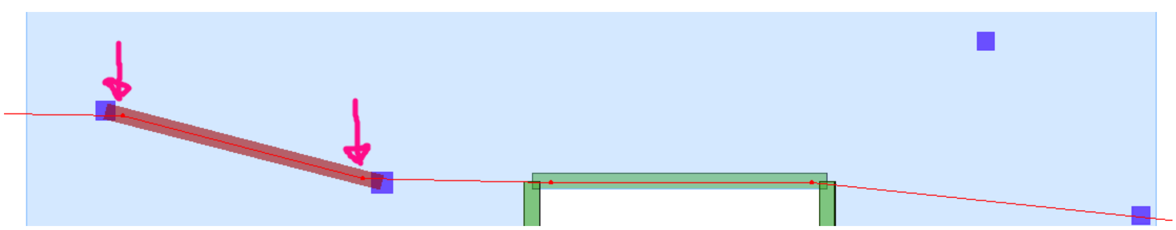
9. If the construction line does not snap on a wall at any location, no vertices are added to the wall.

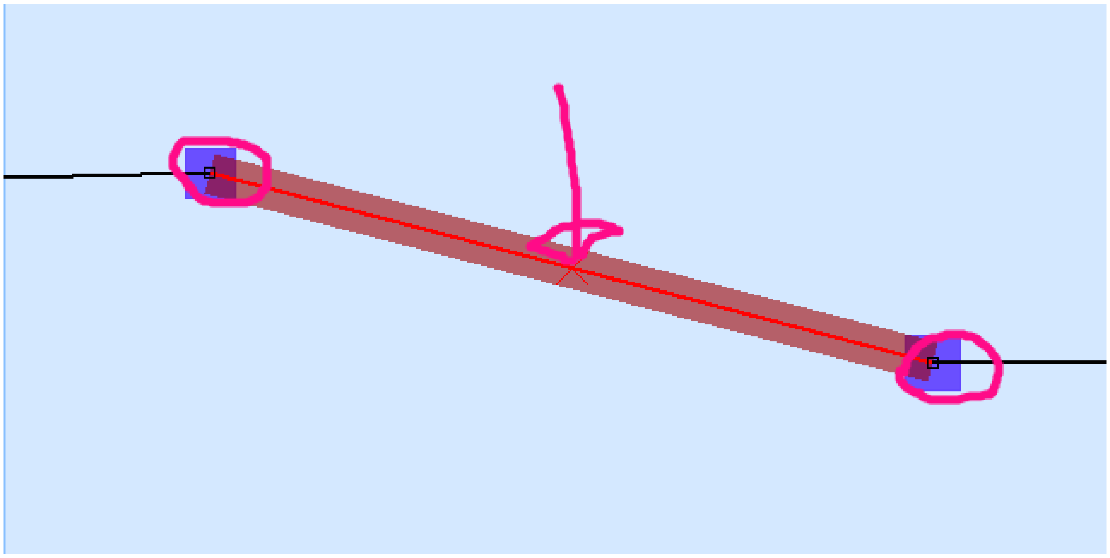
10. For walls oriented perpendicular to a support line or at an angle < 45° relative to a line perpendicular to the support line, it is common to snap a single point at the wall so as to create a "span" where the wall is at the start/end of adjacent spans. See the images below. Note the same rule applies if the snap point is near the wall end, the program places the support line node at the wall endpoint.





11. Placement of construction line snap points are similar to walls. If the snap point is near the end of a beam, the program places the support line nodes at the beam endpoints. In addition the program isolates the support line over the beam and changes the design criteria to 'Beam' criteria.

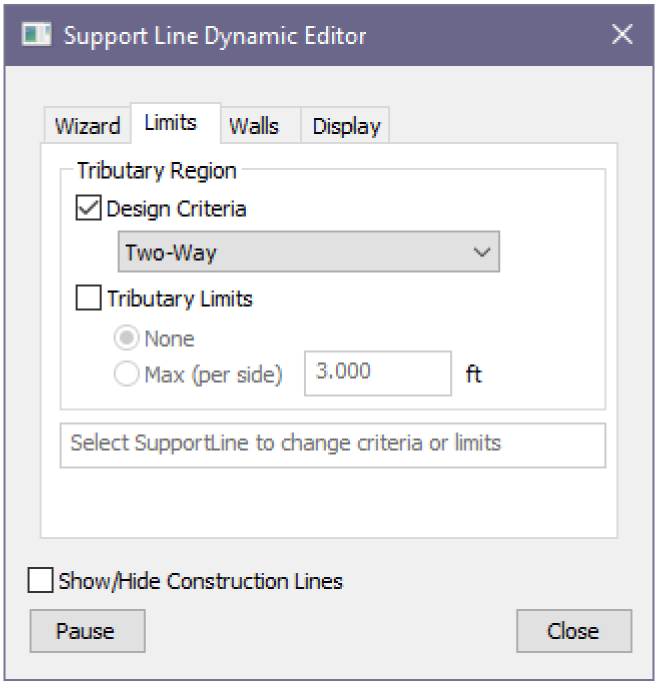




12. For beams oriented perpendicular to a support line or at an angle < 45° relative to a line perpendicular to the support line, it is common to snap a single point at the beam so as to create a "span" where the beam is at the start/end of adjacent spans. Note the same rule applies if the snap point is near the beam end, the program places the support line node at the beam endpoint.

13. The start and end span of a support line created from the wizard follow the same path as the construction line. Therefore, the user may need to re-align the endpoints of the support lines to maintain perpendicularity to the slab edges.

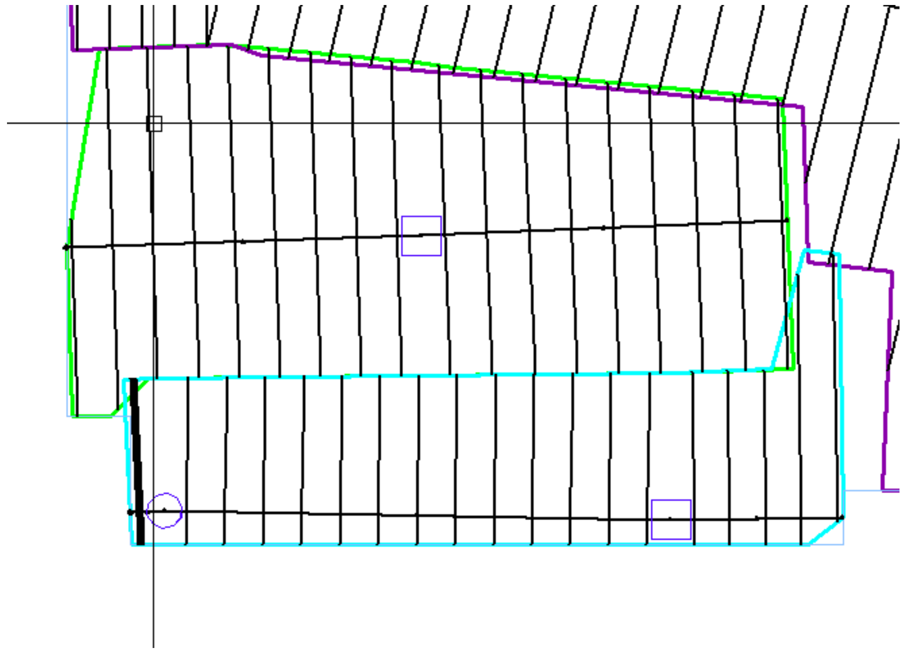
- **Support Line Limits** – Changes selected support line design criteria and tributary limits for the design strip. When this tab is selected, the current design criteria and limits will be displayed on all support lines. Set the desired criteria and snap or window support lines to modify to the desired setting.

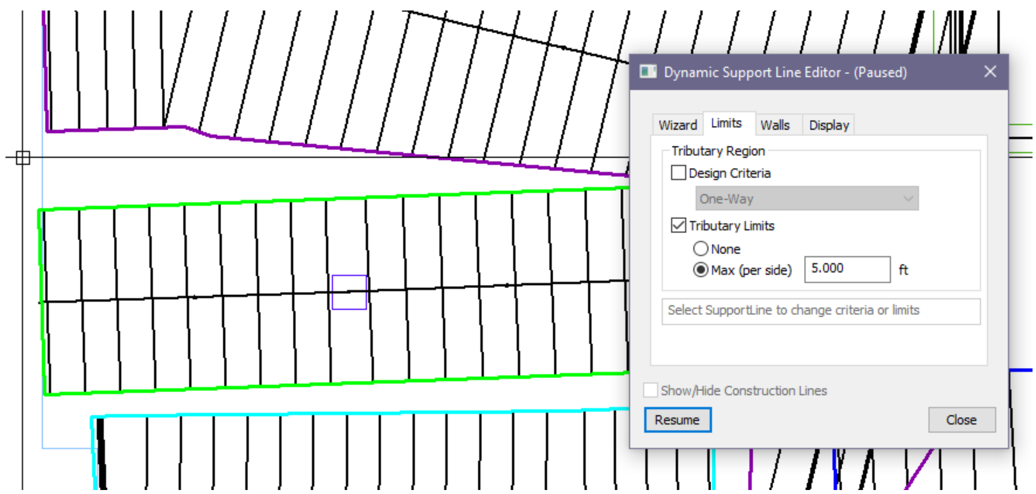


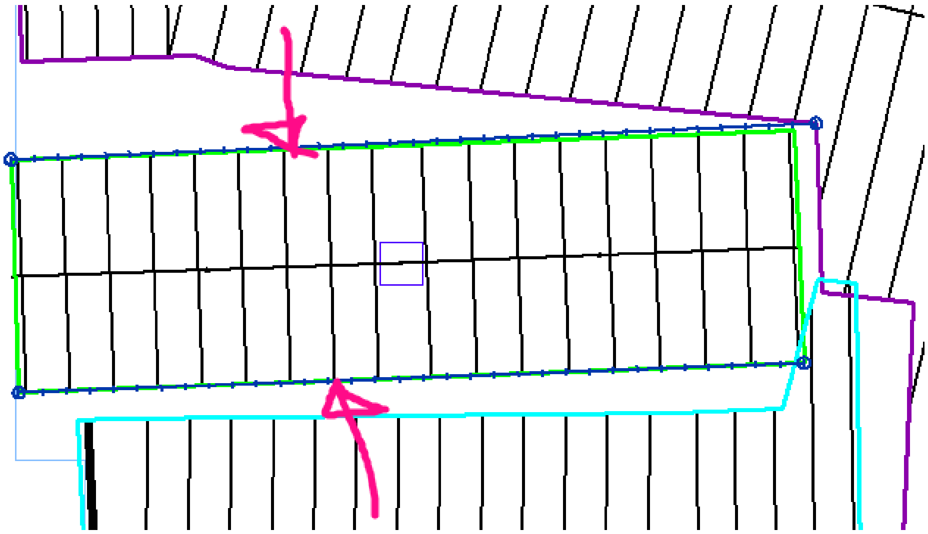
The option to change Tributary Limits allows a user to set the maximum tributary width for a design strip. If 'None' is selected, the program's algorithm will determine the tributary with proximity to adjacent support lines. The image below shows the outcome of the limit set to none for a partial length support line.

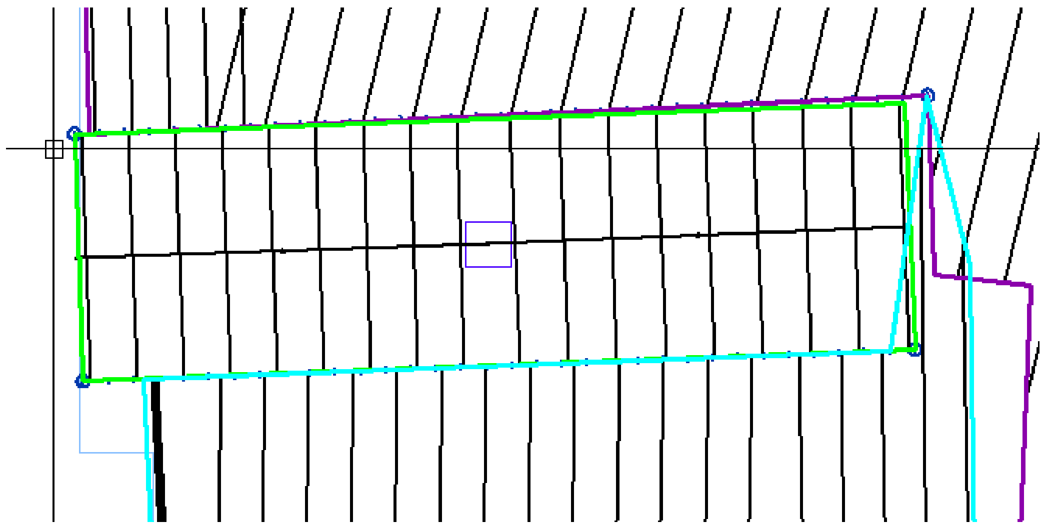
Let us assume we want to limit the tributary to 5' each side and we input this as the limit and regenerate the strips. The 2nd image below displays the outcome. Note that the tributary extents of the design strips above and below don't change. To extend these strips to the limited tributary, splitters can be added. See the 3rd and 4th images below.

Options for tributary limits can also be modified per support line by double-clicking the support line and using 'Design Section Options.'



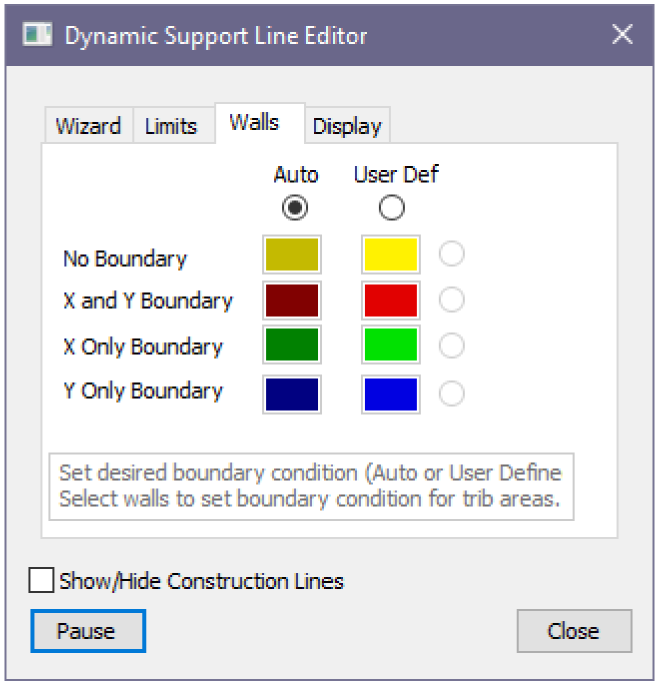

Dynamic Support Line Editor - (Paused)
Wizard
Limits
Walls
Display
Tributary Region
Design Criteria
One-Way
Tributary Limits
None
Max (per side)
5.000
ft
Select SupportLine to change criteria or limits
Show/Hide Construction Lines
Resume
Close



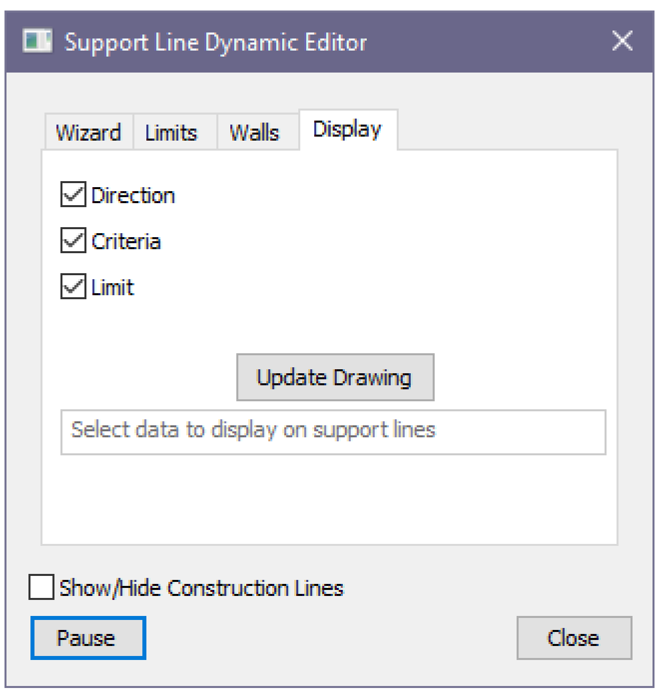


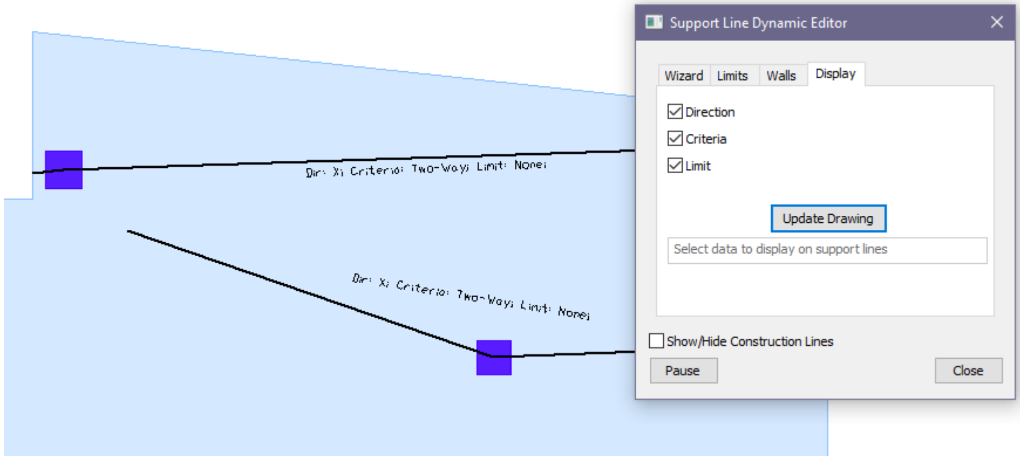
- **Wall** – This tool allows the user to set boundary conditions for how walls are considered for design strip tributary generation. The default conditions are set to automatically consider walls as X and Y boundaries. Options are given for the user to manually define how walls are treated. Select 'User Def' and select the boundary option and then click on the wall to set the boundary. User-defined boundaries remain until the option is set back to 'Auto' or until the manual boundary is set to another option. The link below contains an instructional video that illustrates more detailed use of wall boundaries.

  https://www.youtube.com/watch?v=vbY5_tec9vo



- **Display**- Sets the support line display for Direction, Criteria and Width Limit per the selection of checkboxes. The 'Update Drawing' is used to reflect the current selections and label support lines with those selections. The 2nd image below gives and example of how the information is presented on a support line.

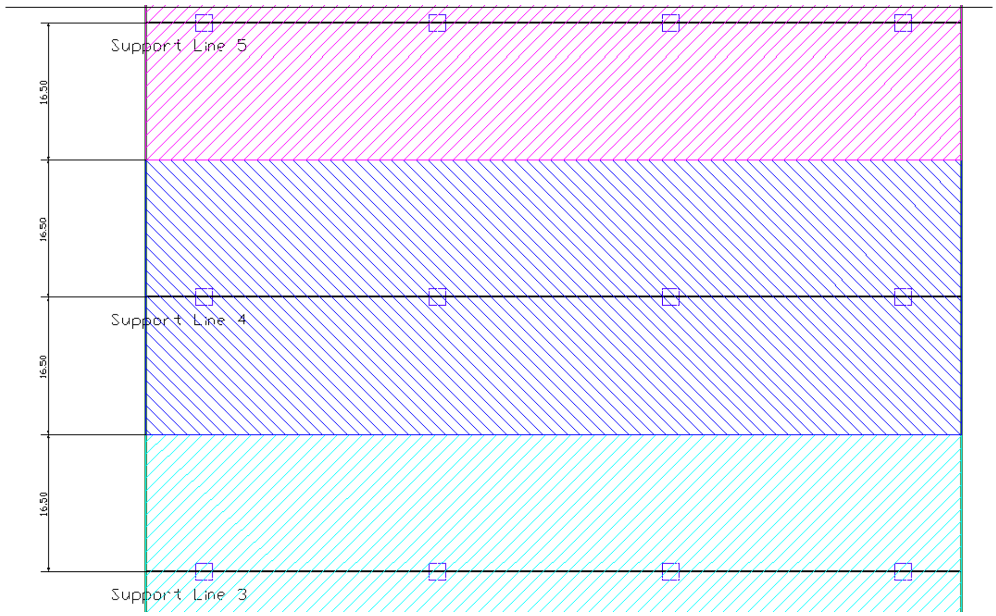


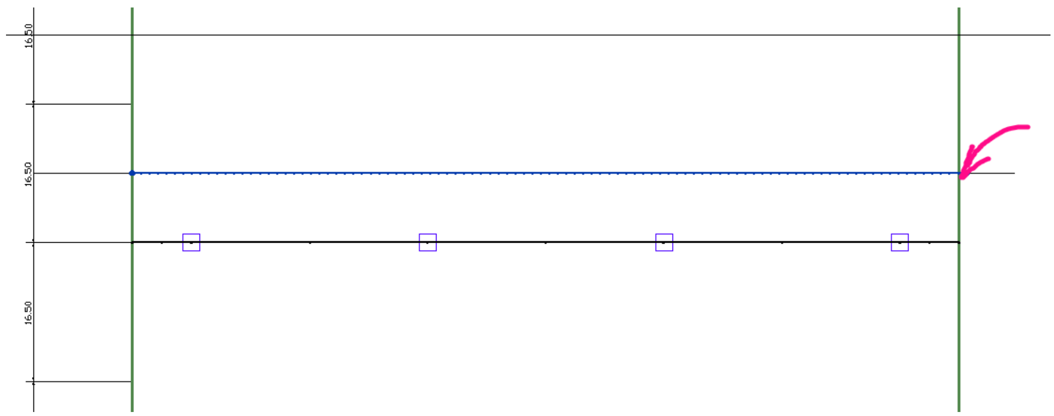


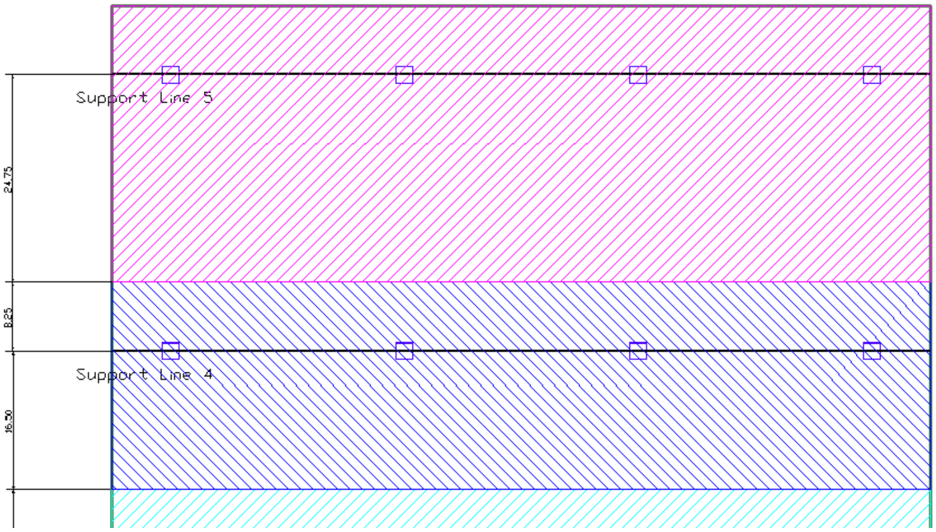
3.2 Splitter functionality

The use of splitters has been simplified in the 2019 version. In earlier versions of the software, the splitter was used to create a virtualized boundary for termination of a tributary region for the design strip. Oftentimes, splitters were required when a support line start and/or end point was located within the perimeter of the slab region. Other cases where splitters were required were limiting tributaries extending through or into openings, support line nodes that shared a common point and one support line was continuous while the other was discontinuous to the shared node.

Splitters were also used to manual generate a boundary that resulted in an uneven distribution of tributary between two adjacent support lines. In the first image below, without use of a splitter, the program generates an even distribution of tributary to Support Lines 3,4 and 5. The bay length is 33' between columns in the Y-direction, therefore the distribution is 16.5' as shown. If a splitter is added 8.25' above Support Line 4, the program treats this as a delimiting boundary for the program-generated design strips. The outcome is shown in the 3rd image below.


Support Line 5
Support Line 4
Support Line 3
16.50
16.50
16.50
16.50

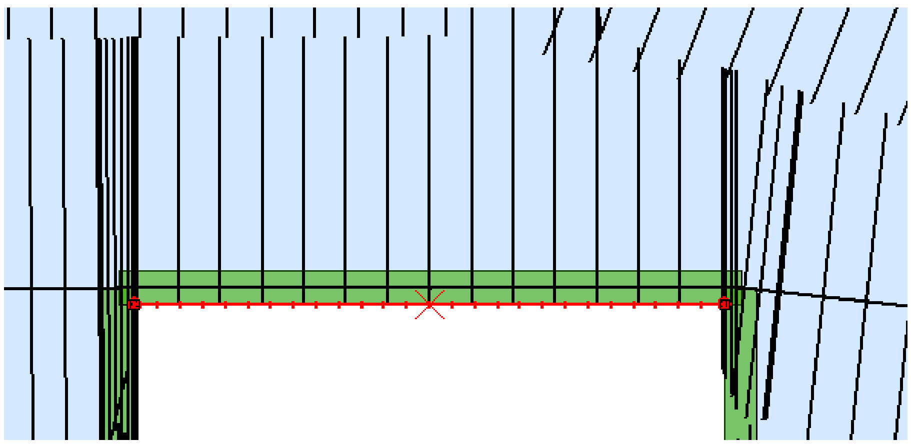

16.50
16.50
16.50


Support Line 5
Support Line 4
24.75
8.25
16.50

New splitter functionality in ADAPT-Builder is approached in a similar manner. Splitters are only required to limited the tributary width of a strip in the direction of the splitter and over the splitter length. In other words, splitters can be used to create a boundary that the tributary edge extends when the width is required to be limited to either side of a support line. Splitters are no longer required for any other purpose as the program has been improved to recognize support line nodes at any location and properly generate strips. Below are a few other cases and descriptions of new splitter use.

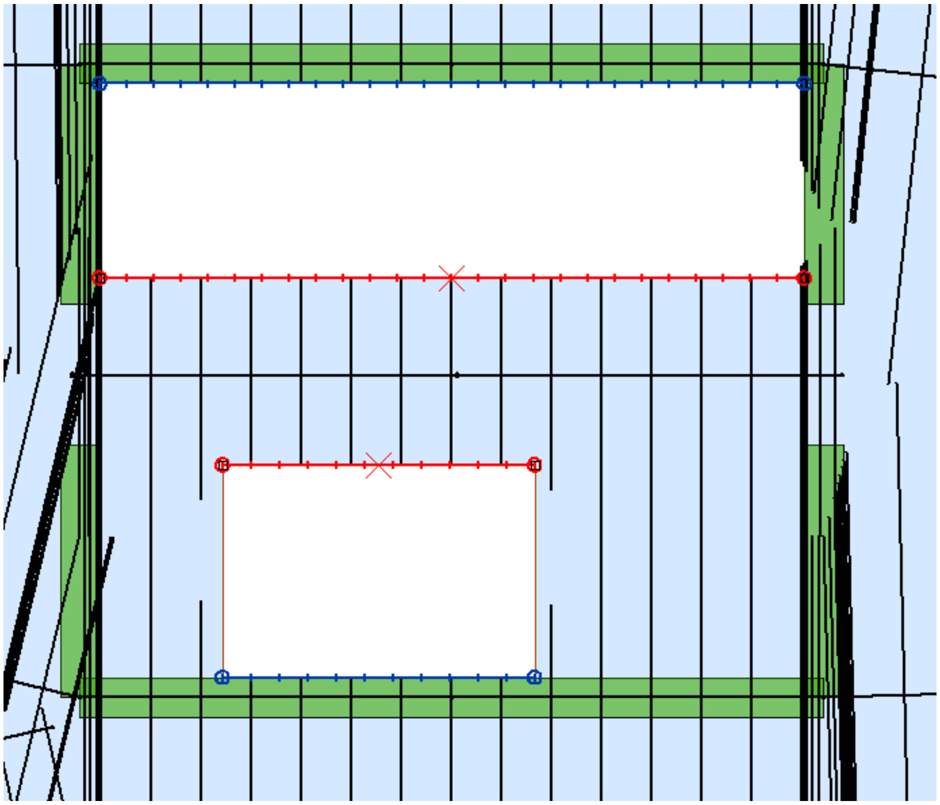
3.2.1 Splitters at Openings

The X-splitters below were added to the side of an opening to limit the tributary from extending into the opening.



3.2.2 Splitters at Slab Regions Between Openings

The X-splitters below were added to both sides of the support line between the openings so as to limit the section cuts extending into the opening.

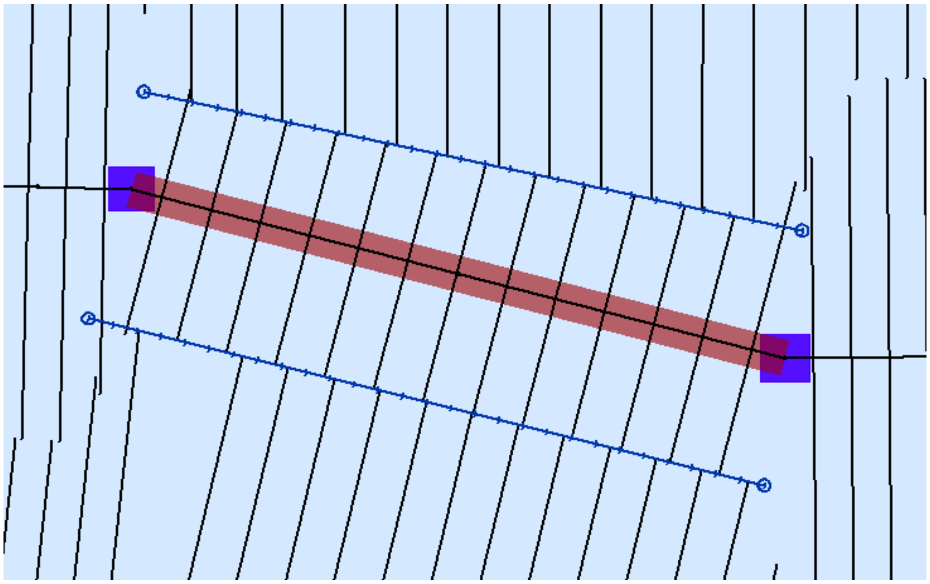


3.2.3 Splitters at Slab or Pit Steps

With the new P-Delta feature, the program graphically reports drift and moment amplification factors (2nd/1st order results) for local axes (RR and SS) as well as the combined

3.2.4 Splitters at beams

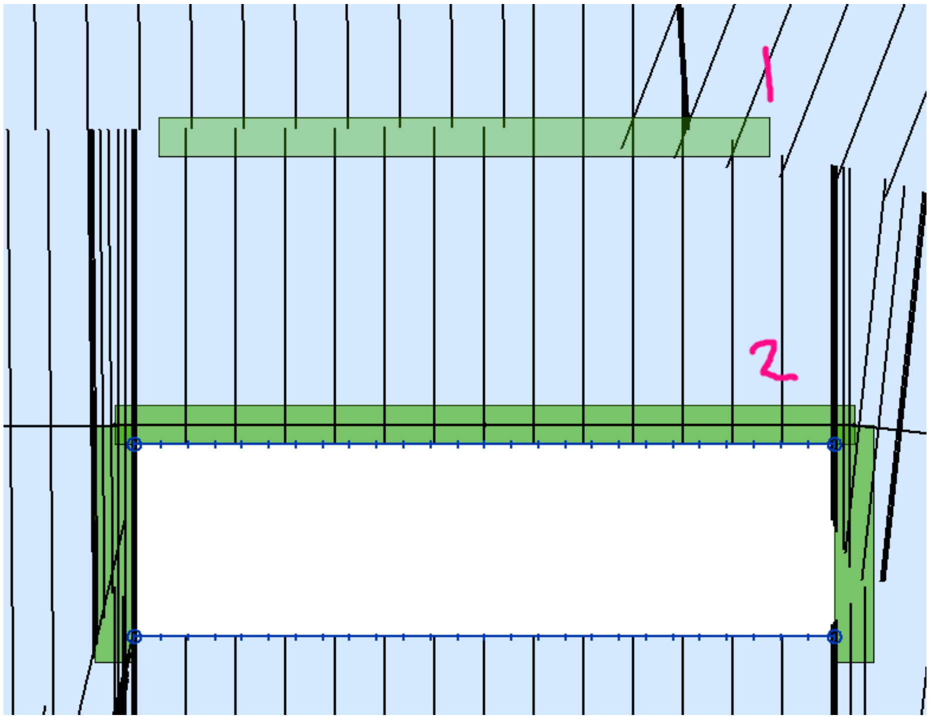
Splitters were added to each side of the beam below to limit the beam flange tributary. This version of the program also includes the option to use the effective flange and properties. If the calculated effective flange to either side is less than the modeled flange (with use of the splitters), the program will use the effective flange in section calculations, otherwise the “physical” tributary and associated properties is used. See Section 2.3 for more information regarding effective flange.

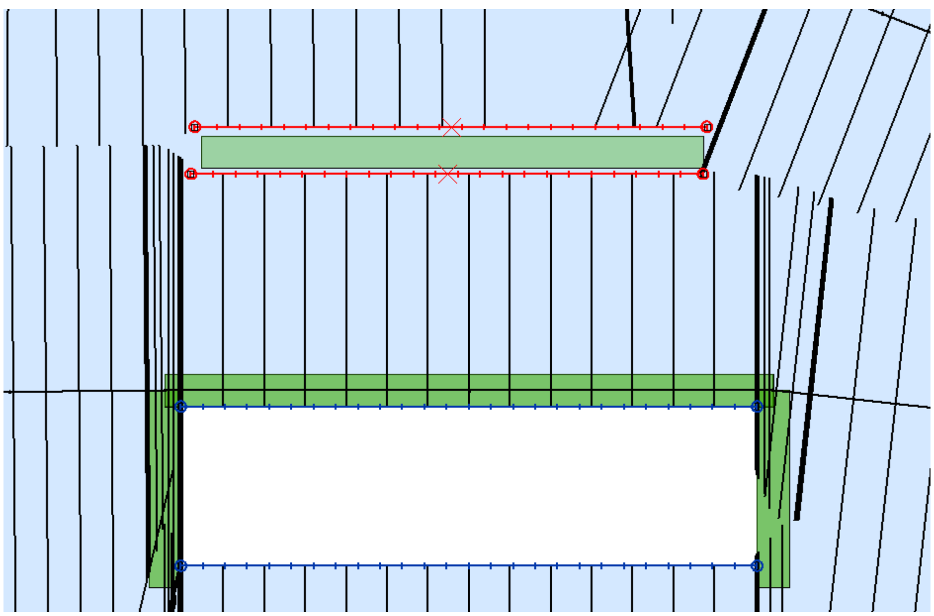


3.2.5 Splitters at walls

Section 3.1 defined the use of a new wall boundary conditions feature. This feature is intended to automatically created boundaries for which a design strip tributary extends to near walls. The use of the boundary condition feature is not required. The user can impose splitters near walls to also manually generate a boundary where the tributary terminates.

In the 1st image below there are walls 1 and 2. Note that the support line in the view is located over wall 2. It is typical for the support line to extend over wall that is oriented in the same direction as the wall is a support for the slab. Wall 1 does not have a support line along its path. Therefore, it is often desirable to extend to the design cuts only to the face of the wall. If a design cut extends over a wall by any length, the program only designs the sections for minimum reinforcement and does not produce a graphical stress result when the wall is part of a two-way slab. The 2nd image below shows the inclusion of splitters and tributary outcome.





The link below contains instructional videos that illustrate more detailed use of splitters.

https://www.youtube.com/watch?v=GZpvL0r5s-I

https://www.youtube.com/watch?v=moznFUczO3Y

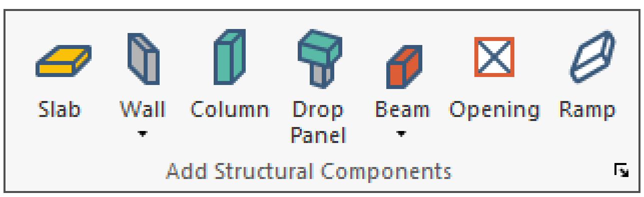
3.3 Ramping

Modeling of ramps is now supported in ADAPT-Builder. Ramps are considered as other analytical components and meshed and utilized FEM solutions. Longitudinal and transverse beams can be modeled at ramps with options to automatically offset the beams as well as offset walls and columns supporting ramps. Ramps are required to be modeled in the same plane and the program constrains the graphical input as 3-point input to enforce planar modeling. Note that the program meshes the ramp as part of pre-processing when the user Analyzes the model. Meshing of slabs using Analysis→Meshing continues to only mesh slabs prior to the analysis. The mesh of ramps and walls is similar in that they are both meshed as part of the general analysis algorithm.

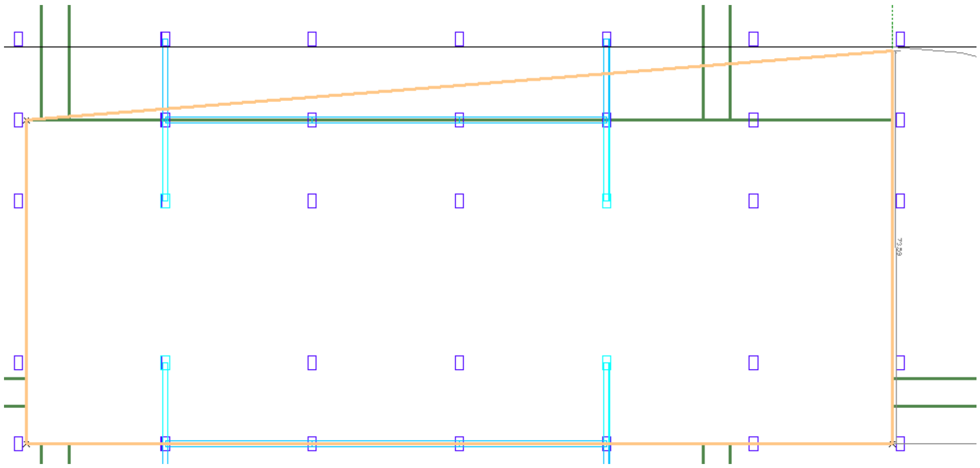
Limitations of ramps in the software include no tendon modeling at ramps, no consideration of ramp elements in the generation of design sections, no design of ramps utilizing design strips and sections, automated or manual. A user can model and offset tendons for beams supporting ramps in the transverse direction.

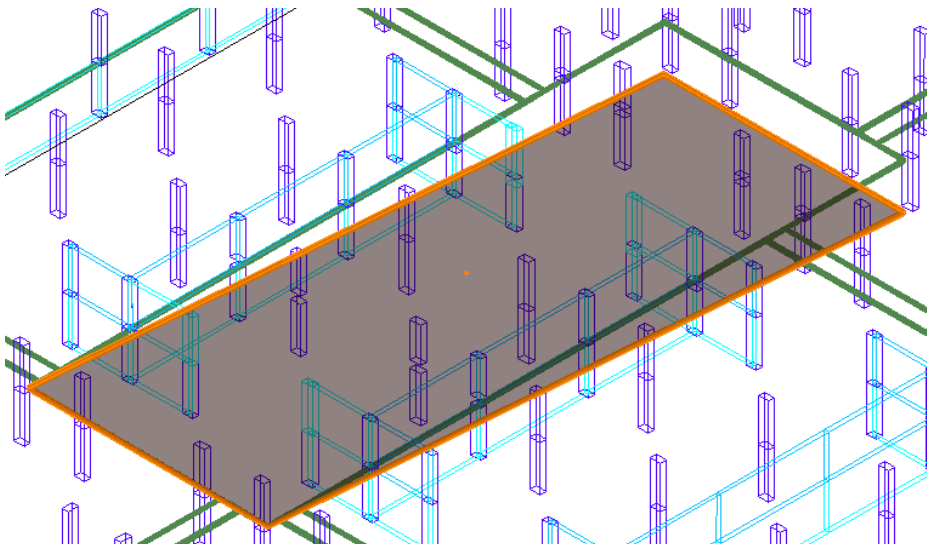
3.3.1 Modeling Ramps

The function used to model ramps is located from Model→Structural Components. When selecting this option, the program will prompt the user to enter the ramp corners. Note that the 1st and 2nd ramp corners define the start of the ramp. The 3rd and 4th corners represent the end (bottom) of the ramp. Note after the 3rd point is selected, the ramp construction line will be locked orthogonally so as to ensure the ramp is planar and not warped.



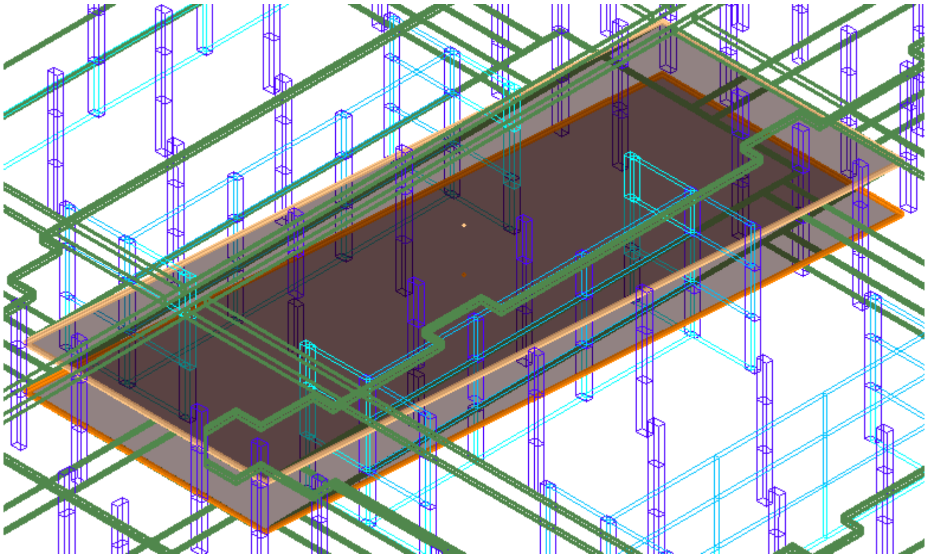
In the image below, the first 3 points are input and the ramp outline is shown in orange color. the start of the ramp is at the left edge and the end of the ramp is at the right edge. The 2nd image below shows a 3d view of the modeled ramp. The slabs shown in the image wrap around the ramp, however, if the slab is located over the ramp, model an opening in the same space.





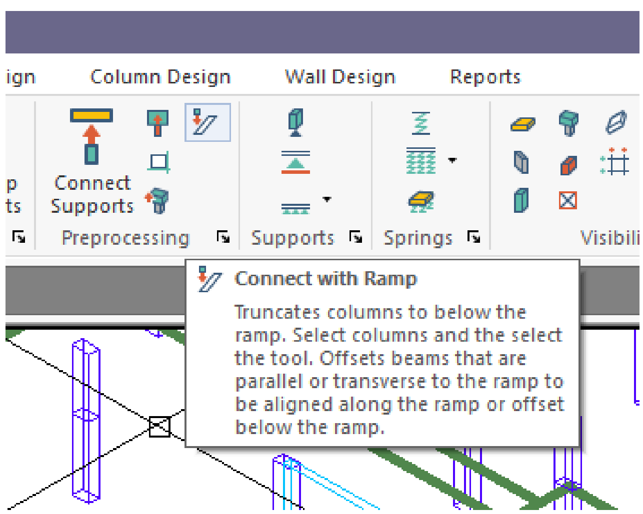
Ramps can be copied or moved vertically in a multistory structure through use of *Modify→Copy/Move→Vertical*. Applying this to the same model, a second ramp is added to the level above as shown below. The user can modify any of the 4 vertices that define a ramp. Upon selecting a point, the ramp entity will auto-adjust so as to maintain the planarity of the surface.

The start and end of a ramp always references a modeled story or plane. If there is a need for the user to model a twisting ramp or ramps at different slopes within the same level, the user is required to created as many planes allow for a multi-sloped or multi-segmented ramp.



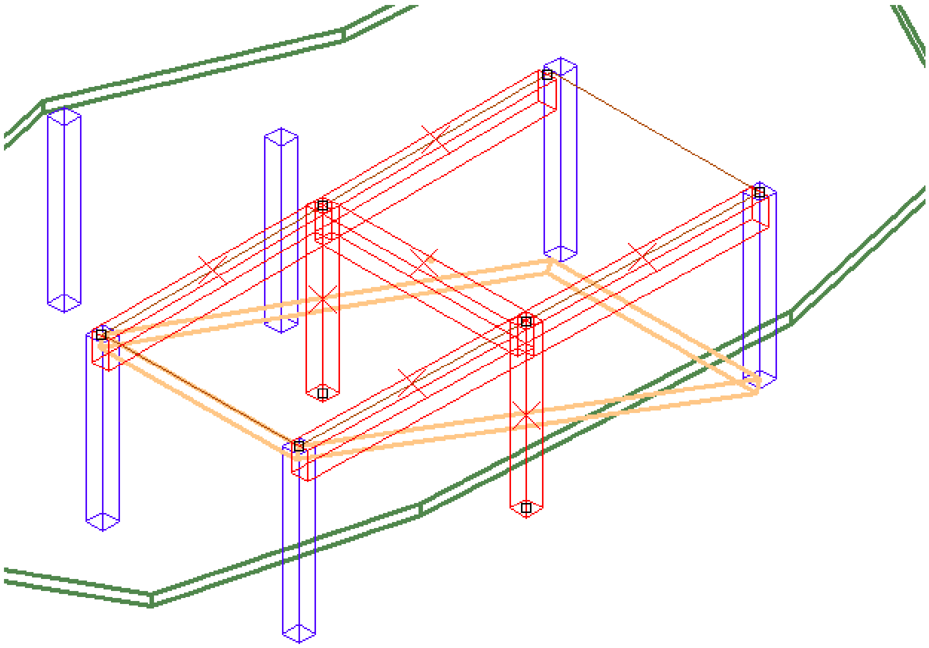
3.3.2 Offsetting Connected Components

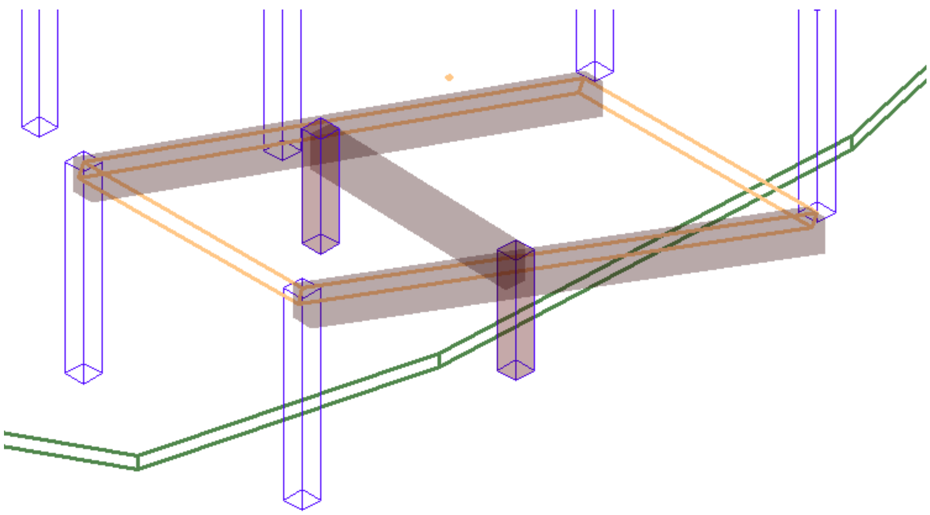
Connected components like beams transverse to the ramp direction, oriented in the same direction as the ramp, or columns located at the edge of a ramp can be automatically offset and aligned with the ramp at their modeled position. To do this, use the tool found from Model→Preprocessing→Connect with Ramp. The component/s required to be offset should be selected first and then use the tool.



In the image below columns and several beams are selected. By default, the offset of the top and bottom (for columns) and start and end (for beams) is set to zero. When using the ‘Connect with

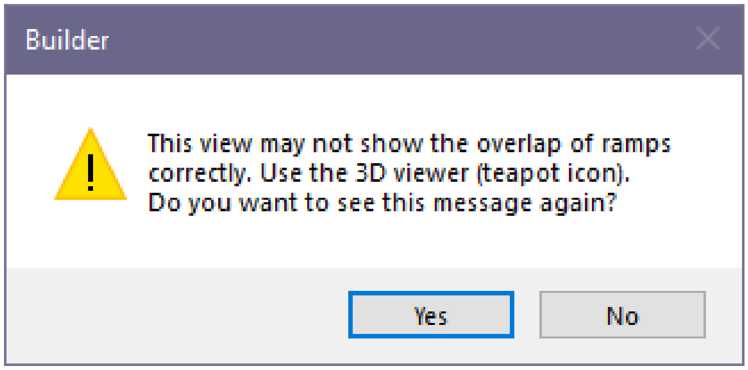
Ramp' tool, the offsets are adjusted to connect the components with the ramp as shown in the 2nd image.



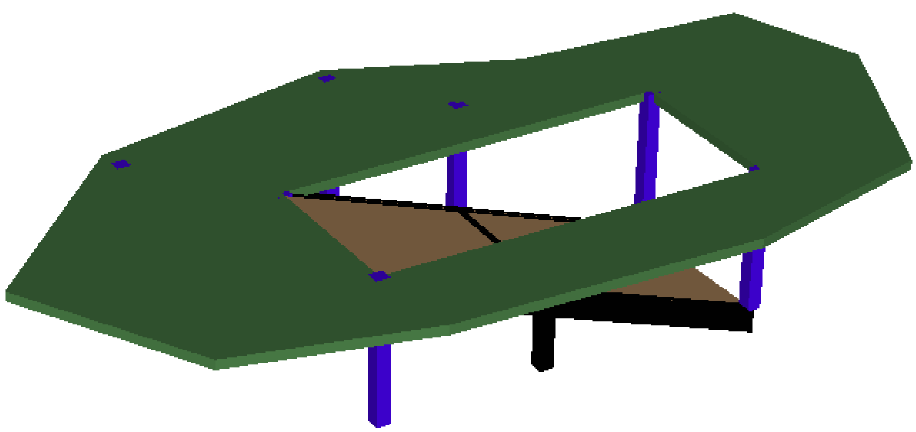


3.3.3 Viewing Options for Ramps

By default, ramps are shown in Top View when modeled. From *Home→Zoom/Camera*, any other 2D view will show the ramp. If the views ‘Top-Front-Right’ or ‘Top-Back-Side’ are invoked, the program will present the following warning.



Selecting ‘Yes’ will continue to the selected 3D view. The user can invoke the rendered view, referenced in the warning, from Visibility→Render Model. The 3D rendered view is shown below with the modeled ramp.



Other tools used for viewing ramps include:

Model→Isolate→Show Ramp Components: Isolates the graphical view to show only ramps in the model. Use Visibility→Default Display, Visibility→View Settings, or Model→Visibility to restore the view of other components.

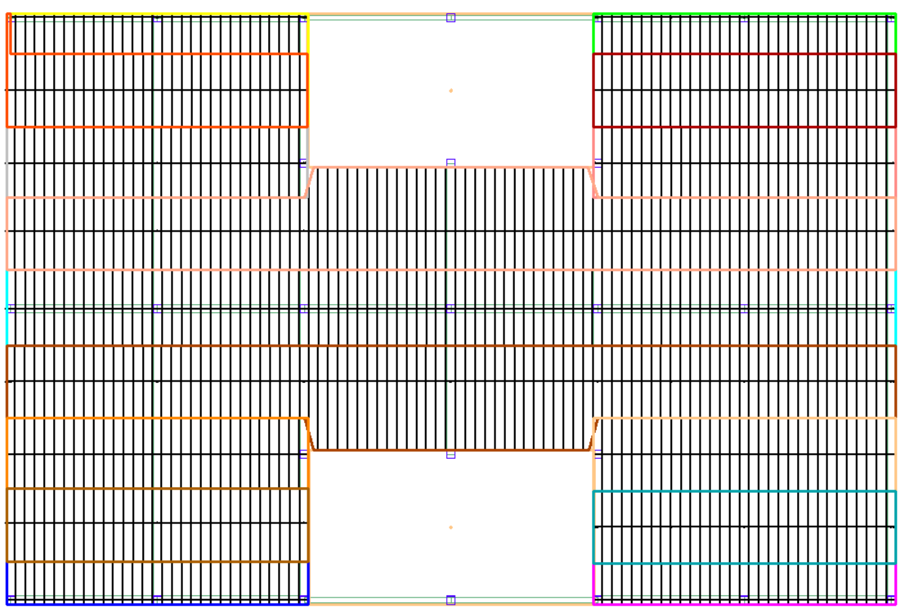
Visibility→View Settings→Structural Components: This displays On/Off the ramps in the model view.

Model→Visibility→Ramps: This displays On/Off the ramps in the model view.

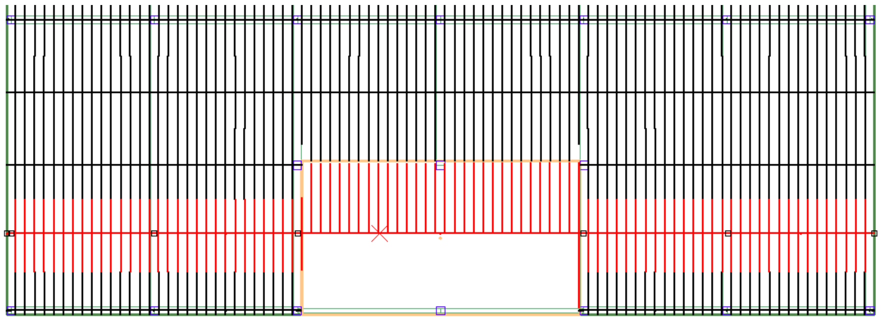
3.3.4 Design Strips and Splitters near Ramps

The current implementation of ramps in ADAPT-Builder does not include the design of sections cutting through ramps. In other words, while a physical design cut may intersect a ramp, the ramp itself and associated offsets are not included in the idealized design section that the program processes for design of floor sections.

It is recommended to model design strip such that the design cuts do not extend into ramp regions. This will aid the user in avoiding confusion and clearly demarking that portion of the slab that the program properly handles for design. The image below shows an example of ramps at the top-center and bottom-center of the slab region. In this example, the support lines are modeled to the start and end edge of the ramp. Note the program treats the long, interior edges of the ramp as a boundary and the design strip does not extend into the ramp. The user can also add splitters to limit an outcome where a tributary extends into the ramp space.

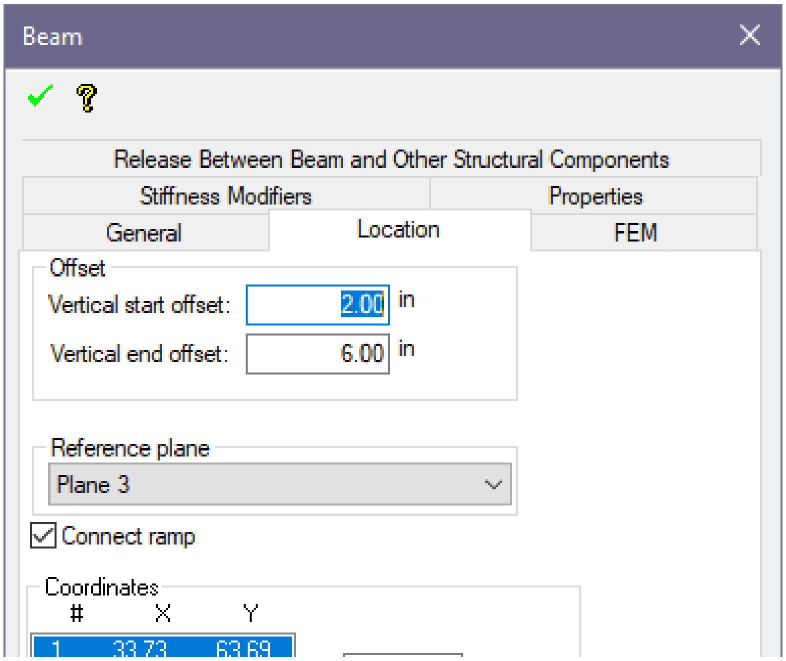


In the image shown below, notice the support line was modified to extend continous through the ramp. This being improper and non-recommended input leads to a tributary portion including ½ of the ramp. In this scenario, even if a tributary is created in this ramp space, when the design sections are generated and design for this strip, the sections in the ramp are automatically removed.



3.4 Beam End Offsets

Beam coordinate definitions now include the ability to create offsets at the beam start and end points. This was a necessary improvement to support inclined beams at edges of ramps. Beams not associated with ramps can also be offset with different values at each end. Double-click on a beam to open the beam properties and use the 'Location' tab to enter the offset. If the option 'Connect Ramp' is selected, different offsets can be modeled. If this button is not selected, the program will use the same offset at both ends.



Chapter 4

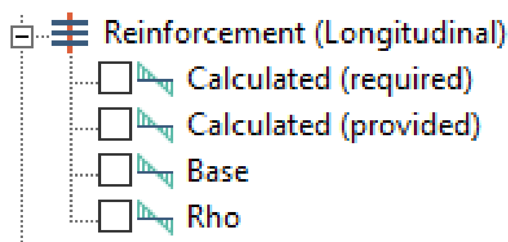
REPORTING IMPROVEMENTS

4. REPORTING IMPROVEMENTS

4.1 Graphical Longitudinal Reinforcement on Design Strips

The Result Browser includes new graphical reporting options for design sections reporting the area of steel for provided and required reinforcement per program calculation and base (user-defined) reinforcement. In addition, the value of 'rho' is reported as the total area of steel (base + provided) divided by the section area.

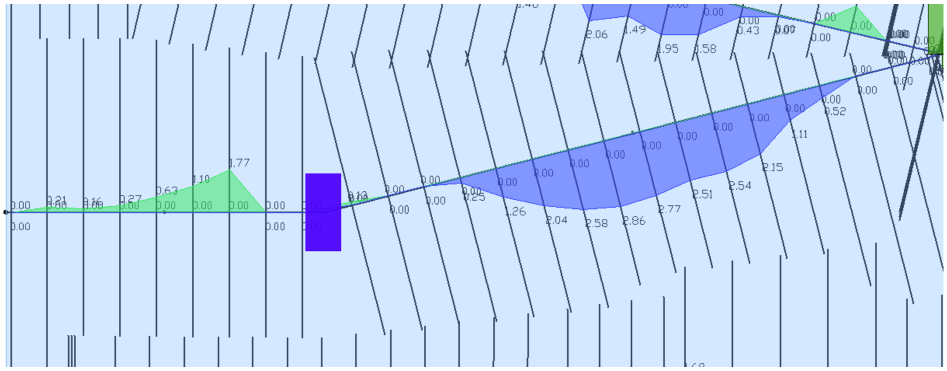
These new options are available after the design of sections has been completed from Floor Design→Section Design→Design/Investigate the Sections. Note if the option to 'Investigate' is used, the 'Calculated' options below will report zero on design strips as these are intended for the 'Design' option. These options shown below are found in the 'Analysis' tab of the Results Browser from Design Sections-Reinforcement (longitudinal).



4.1.1 Calculated (required)

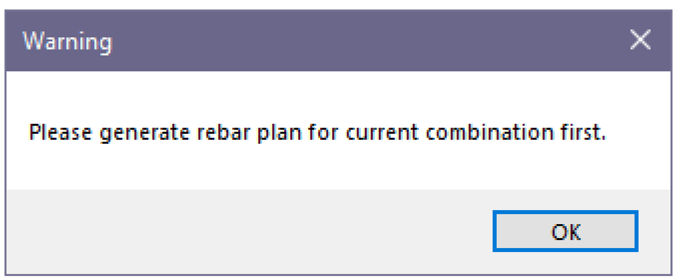
Selecting this option will report the total reinforcement required as calculated rebar at the top and bottom of the design sections. The graphical presentation is shown for the combination or Envelope selected from 'Loads' tab in the Results Browser. It is not required to produce the rebar plan prior to selecting this option.

The image below shows a presentation of the output. The green graph reports the required top reinforcement and the blue graph reports the required bottom reinforcement. The values are shown in in2 and mm2 as defaults, depending on the selected unit system.



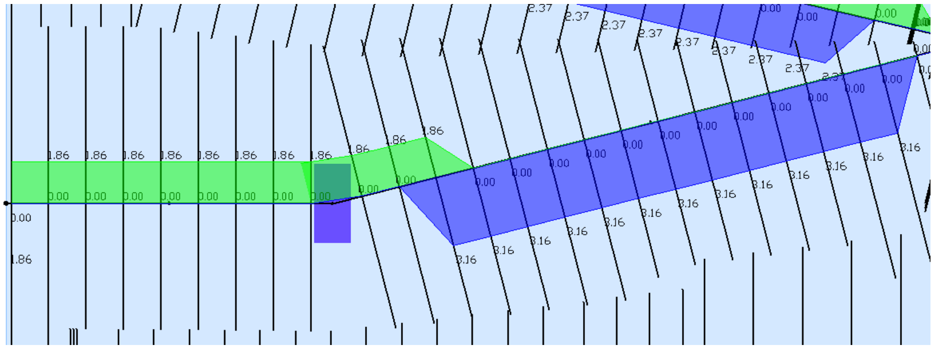
4.1.2 Calculated (provided)

Selecting this option will report the total reinforcement provided as calculated rebar at the top and bottom of the design sections. This is a function of the user-defined bar sizes for the system criteria type for top and bottom longitudinal bars. These options can be defined from Criteria→Rebar Size/Material. The graphical presentation is shown for the combination or Envelope selected from ‘Loads’ tab in the Results Browser. It is required to produce the rebar plan prior to selecting this option. If the rebar plan for the selected Load Combination or envelope type has not been produced, a warning will appear as shown below.



The rebar plan is produced from Floor Design→Rebar→Calculated Rebar Plan or from Rebar→Generate→Calculated Rebar Plan.

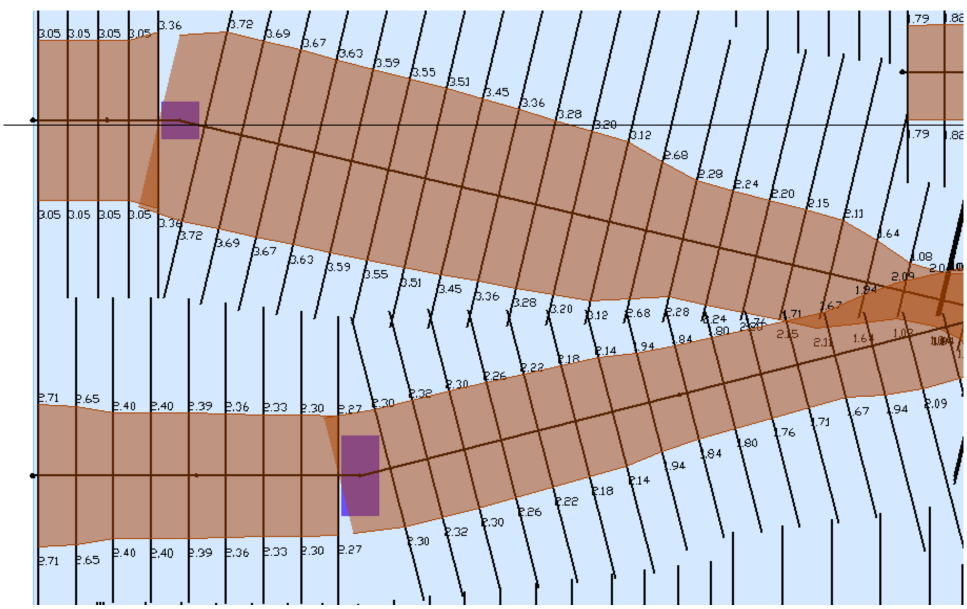
The image below shows a presentation of the output. The green graph reports the provided top reinforcement and the blue graph reports the provided bottom reinforcement. The values are shown in in2 and mm2 as defaults, depending on the selected unit system.



4.1.3 Base

Selecting this option will report the total base (user-defined) reinforcement modeled in the slab and/or beam at the top and bottom of the design sections. The user must Design or Investigate the sections prior to producing the result. The graphical presentation is shown for the combination or Envelope selected from 'Loads' tab in the Results Browser.

The image below shows a presentation of the output. Both top and bottom graphs are reported as orange. The values are shown in in2 and mm2 as defaults, depending on the selected unit system.



4.1.4 Rho

Selecting this option will report total area of base rebar plus calculated required rebar for the selected load combination, divided by the total area of the design section. Prestressing steel area is not included in this calculation. The user must Design or Investigate the sections prior to producing the result. The graphical presentation is shown for the combination or Envelope selected from 'Loads' tab in the Results Browser. The value presented is shown as %.

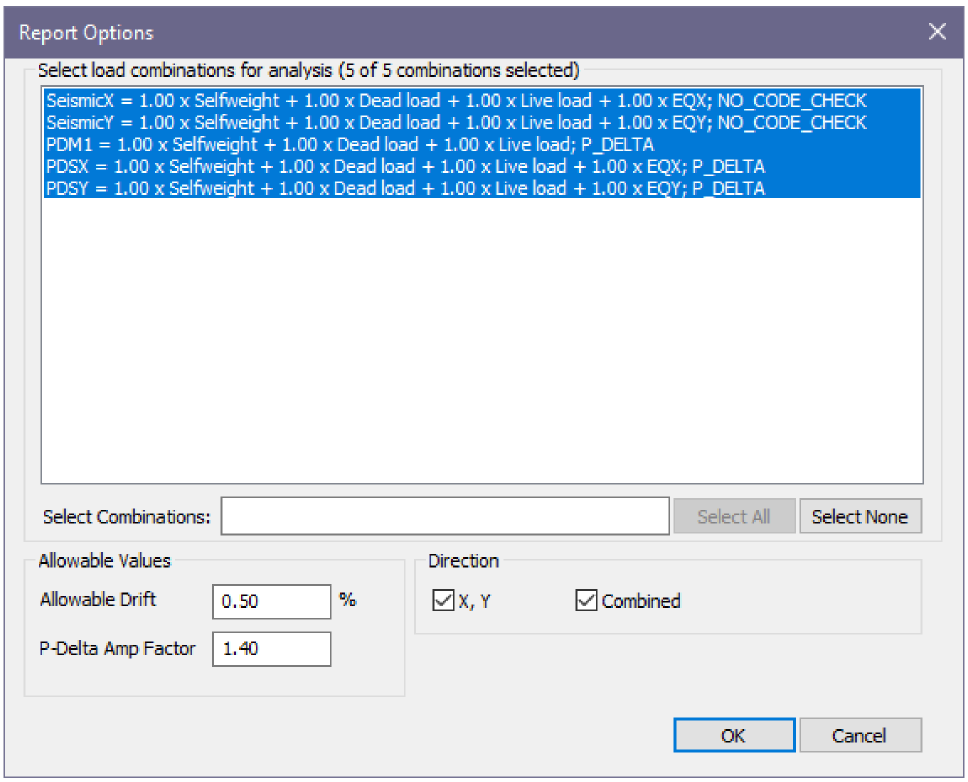
4.2 XLS File Reporting for Drift Results

The program now includes the option to produce a .XLS report with multiple data sheets for lateral drift results. These results can include or exclude P-Delta combinations or only 1st order lateral results. The new reporting feature is invoked from Reports→Analysis Data→XLS Reports→Drift Report. Prerequisite to producing the report, the model should be run in Multi-Level mode and contain at least a single combination with lateral loads included, although this is not required.

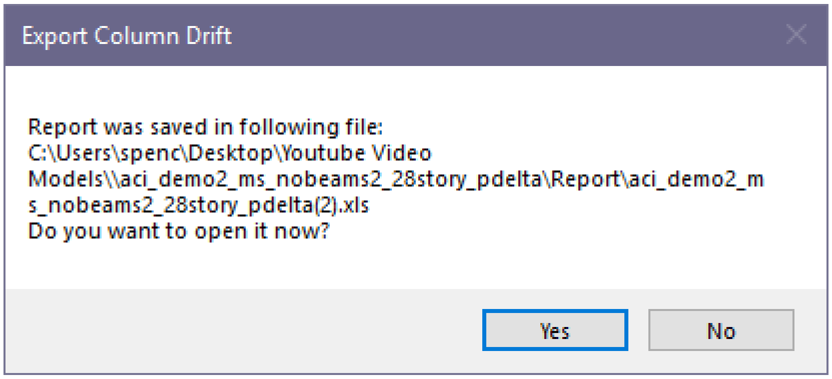
When you select the option to produce the drift report, the following dialog window will appear, allowing the user to customize the report. This dialog includes a list of design combinations for which a global FEM solution has been generated. The user can custom-select which combination/s to produce the report for.

In addition, the user can enter the allowable drift % as well as the P-Delta Maximum Drift Amplification Factor to check against. Note that the allowable drift is that entered by the user. **It is the users responsibility to apply any displacement amplification factor (Cd) within the value entered as allowable**. Note these two inputs within this dialog are separate and do not affect those located in the 'Settings' tab of the Result Browser, which are intended for graphical code check of drift and amplification due to P-Delta.

The user can also select which direction to produce the report for. Options include global X and Y as well as the combined drift.



Upon selection OK, the program will produce a message showing the path location ot the XLS file and prompt the user to open the file automatically. The file will open to the default application compatible with XLS files. This is typically Microsoft Excel® or other equivalent application.



The XLS file will generate 7 unique sheets of data. These include Stations, Displacements, Drift at Columns, Average Drift, Max Drift Envelope, Average Drift Envelope, and Drift Amplification Factor. Each sheet is described briefly below with an illustrative example.

4.2.1 Stations

This report sheet lists all columns in the model starting at the upper-most plane and working down. The column label, global coordinates and height are shown.

	A	B	C	D	E	F
1	Ref plane	Station	X	Y	Z	Height
2			ft	ft	ft	ft
3	Plane 28	Column 251	133.694	81.005	285.000	10.000
4	Plane 28	Column 252	133.798	53.537	285.000	10.000
5	Plane 28	Column 253	133.444	33.456	285.000	10.000
6	Plane 28	Column 254	132.694	11.381	285.000	10.000
7	Plane 28	Column 255	106.190	9.347	285.000	10.000
8	Plane 28	Column 256	97.449	79.799	285.000	10.000
9	Plane 28	Column 257	77.845	7.614	285.000	10.000
10	Plane 28	Column 281	76.120	31.406	285.000	10.000
11	Plane 27	Column 244	133.694	81.005	275.000	10.000
12	Plane 27	Column 245	133.798	53.537	275.000	10.000
13	Plane 27	Column 246	133.444	33.456	275.000	10.000

4.2.2 Displacements

This report sheet shows the displacements for all columns at the selected coordinates for the top and bottom location of the column. The selected combinations for the produces results are listed along with the combination analysis/design option type. The columns are reported by plane listed top-down.

	A	B	C	D	E	F	G	H	I	J
1	Ref plane	Station	LC	Type	Xtop	Xbott	Ytop	Ybott	Combined top	Combined bott
2					in	in	in	in	in	in
3	Plane 28	Column 251	PDSX	P_Delta	7.43	6.71	10.19	10.36	12.61	12.34
4	Plane 28	Column 251	PDSY	P_Delta	-2.14	-2.39	37.40	36.70	37.46	36.77
5	Plane 28	Column 252	PDSX	P_Delta	19.99	19.13	9.73	9.92	22.24	21.55
6	Plane 28	Column 252	PDSY	P_Delta	6.36	6.06	37.09	36.40	37.63	36.90
7	Plane 28	Column 253	PDSX	P_Delta	29.16	28.20	9.56	9.76	30.69	29.84
8	Plane 28	Column 253	PDSY	P_Delta	12.55	12.23	36.97	36.29	39.05	38.29
9	Plane 28	Column 254	PDSX	P_Delta	39.26	38.17	9.23	9.42	40.33	39.32
10	Plane 28	Column 254	PDSY	P_Delta	19.37	19.01	36.74	36.05	41.54	40.76
11	Plane 28	Column 255	PDSX	P_Delta	40.21	39.10	-2.91	-2.56	40.31	39.18
12	Plane 28	Column 255	PDSY	P_Delta	20.01	19.64	28.55	27.91	34.86	34.13
13	Plane 28	Column 256	PDSX	P_Delta	8.00	7.26	-6.89	-6.51	10.56	9.75
14	Plane 28	Column 256	PDSY	P_Delta	-1.75	-2.02	25.87	25.21	25.93	25.29
15	Plane 28	Column 257	PDSX	P_Delta	41.01	39.88	-15.90	-15.37	43.98	42.74

4.2.3 Drift at Columns

This report shows the inter-story drift values (in or mm) for each column, at each plane for the selected combinations and directions. The combination analysis design/design option type is listed.

	A	B	C	D	E	F	G
1	**Ref plane**	**Station**	**LC**	**Type**	**Drift X**	**Drift Y**	**Drift Combined**
2					in	in	in
3	Plane 28	Column 251	PDSX	P_Delta	0.72	0.17	0.74
4	Plane 28	Column 251	PDSY	P_Delta	0.26	0.70	0.75
5	Plane 28	Column 252	PDSX	P_Delta	0.87	0.19	0.89
6	Plane 28	Column 252	PDSY	P_Delta	0.30	0.69	0.75
7	Plane 28	Column 253	PDSX	P_Delta	0.97	0.19	0.98
8	Plane 28	Column 253	PDSY	P_Delta	0.32	0.69	0.76
9	Plane 28	Column 254	PDSX	P_Delta	1.09	0.19	1.11
10	Plane 28	Column 254	PDSY	P_Delta	0.36	0.69	0.78
11	Plane 28	Column 255	PDSX	P_Delta	1.11	0.35	1.16
12	Plane 28	Column 255	PDSY	P_Delta	0.37	0.64	0.74
13	Plane 28	Column 256	PDSX	P_Delta	0.74	0.38	0.83
14	Plane 28	Column 256	PDSY	P_Delta	0.27	0.66	0.71
15	Plane 28	Column 257	PDSX	P_Delta	1.13	0.53	1.25
16	Plane 28	Column 257	PDSY	P_Delta	0.38	0.57	0.69
17	Plane 28	Column 281	PDSX	P_Delta	0.99	0.55	1.13

Additional results shown on this sheet are for code check against the maximum allowable drift %. The % drift for the selected directions are shown as well as the allowable %. If any of the direction values (X, Y or combined) exceed the allowable the program will flag this row (indicating a single column for a single load combination result) as **NG**. If the values are less than allowable the row will be flagged as **OK**.

	G	H	I	J	K	L
	Drift Combined	Drift X	Drift Y	Drift Combined	Allowable	Status
	in	%	%	%	%	
17	0.74	0.60	0.14	0.61	0.50	NG
70	0.75	0.21	0.59	0.62	0.50	NG
19	0.89	0.72	0.16	0.74	0.50	NG
59	0.75	0.25	0.58	0.63	0.50	NG
19	0.98	0.80	0.16	0.82	0.50	NG
59	0.76	0.27	0.57	0.63	0.50	NG
19	1.11	0.91	0.16	0.92	0.50	NG
59	0.78	0.30	0.58	0.65	0.50	NG
35	1.16	0.93	0.29	0.97	0.50	NG
54	0.74	0.31	0.53	0.62	0.50	NG
38	0.83	0.62	0.32	0.69	0.50	NG
56	0.71	0.23	0.55	0.59	0.50	NG
53	1.25	0.94	0.44	1.04	0.50	NG
57	0.69	0.32	0.48	0.57	0.50	NG
55	1.13	0.83	0.46	0.95	0.50	NG

4.2.4 Average Drift

This report shows the average inter-story drift values (in or mm) for all columns at a plane and for each selected combination and direction. The combination analysis design/design option type is listed. To produce the values, the program sums the drift for all columns and divides by the number of columns. This is done for each direction and combination.

	A	B	C	D	E	F
1	Ref plane	LC	Type	Drift X	Drift Y	Drift Combined
2				in	in	in
3	Plane 28	PDSX	P_Delta	0.95	0.32	1.01
4	Plane 28	PDSY	P_Delta	0.32	0.65	0.73
5	Plane 27	PDSX	P_Delta	0.51	0.15	0.54
6	Plane 27	PDSY	P_Delta	0.17	0.36	0.40
7	Plane 26	PDSX	P_Delta	0.52	0.13	0.55
8	Plane 26	PDSY	P_Delta	0.18	0.39	0.44
9	Plane 25	PDSX	P_Delta	0.54	0.12	0.57
10	Plane 25	PDSY	P_Delta	0.19	0.42	0.47
11	Plane 24	PDSX	P_Delta	0.56	0.14	0.59
12	Plane 24	PDSY	P_Delta	0.21	0.46	0.52
13	Plane 23	PDSX	P_Delta	0.57	0.16	0.61
14	Plane 23	PDSY	P_Delta	0.22	0.50	0.56
15	Plane 22	PDSX	P_Delta	0.59	0.18	0.63
16	Plane 22	PDSY	P_Delta	0.23	0.54	0.60
17	Plane 21	PDSX	P_Delta	0.61	0.20	0.65

Additional results shown on this sheet are for code check against the maximum allowable drift %. The % drift for the selected directions are shown as well as the allowable %. If any of the direction values (X, Y or combined) exceed the allowable the program will flag this row (indicating a single column for a single load combination result) as **NG**. If the values are less than allowable the row will be flagged as **OK**. Note that for each plane and combination, the program will **bold** the text for those drift % values that exceed the allowable.

F	G	H	I	J	K
Drift Combined	Drift X	Drift Y	Drift Combined	Allowable	Status
in	%	%	%	%	
1.01	**0.79**	0.27	**0.84**	0.50	NG
0.73	0.27	**0.54**	0.61	0.50	NG
0.54	**0.42**	0.12	**0.45**	0.50	OK
0.40	0.14	**0.30**	0.34	0.50	OK
0.55	**0.44**	0.11	**0.46**	0.50	OK
0.44	0.15	**0.32**	0.36	0.50	OK
0.57	**0.45**	0.10	**0.47**	0.50	OK
0.47	0.16	**0.35**	0.40	0.50	OK
0.59	**0.46**	0.12	**0.49**	0.50	OK
0.52	0.17	**0.38**	0.43	0.50	OK
0.61	**0.48**	0.14	**0.50**	0.50	OK
0.56	0.18	**0.42**	0.47	0.50	OK
0.63	**0.49**	0.15	**0.52**	0.50	NG
0.60	0.19	**0.45**	0.50	0.50	OK
0.65	**0.50**	0.17	**0.54**	0.50	NG

4.2.5 Maximum Drift Envelope

This sheet reports the maximum drift % against the maximum allowable drift % for all combinations and all columns at a plane. The % drift for the selected directions are shown as well as the allowable %. If any of the direction values (X, Y or combined) exceed the allowable the program will flag this row (indicating a single column for a single load combination result) as **NG**. If the values are less than allowable the row will be flagged as **OK**.

	A	B	C	D	E	F	G
1	Ref plane	Elevation	Drift X	Drift Y	Drift Combined	Allowable	Status
2		ft	%	%	%	%	
3	Plane 28	285.00	0.94	0.59	1.04	0.50	NG
4	Plane 27	275.00	1.05	0.68	1.15	0.50	NG
5	Plane 26	265.00	1.12	0.75	1.22	0.50	NG
6	Plane 25	255.00	1.18	0.83	1.29	0.50	NG
7	Plane 24	245.00	1.26	0.91	1.36	0.50	NG
8	Plane 23	235.00	1.33	1.00	1.44	0.50	NG
9	Plane 22	225.00	1.41	1.09	1.52	0.50	NG
10	Plane 21	215.00	1.48	1.18	1.59	0.50	NG
11	Plane 20	205.00	1.54	1.26	1.66	0.50	NG
12	Plane 19	195.00	1.61	1.34	1.73	0.50	NG
13	Plane 18	185.00	1.66	1.42	1.79	0.50	NG
14	Plane 17	175.00	1.71	1.48	1.84	0.50	NG
15	Plane 16	165.00	1.76	1.54	1.88	0.50	NG
16	Plane 15	155.00	1.79	1.60	1.92	0.50	NG
17	Plane 14	145.00	1.82	1.64	1.94	0.50	NG

4.2.6 Average Drift Envelope

This sheet reports the maximum average drift % against the maximum allowable drift % for all combinations and all columns at a plane. The % drift for the selected directions are shown as well as the allowable %. If the any of the direction values (X, Y or combined) exceed the allowable the program will flag this row (indicating a single column for a single load combination result) as **NG**. If the values are less than allowable the row will be flagged as **OK**. Note that the program will **bold** the text for those drift % values that exceed the allowable.

Ref plane	Elevation	Drift X	Drift Y	Drift Combined	Allowable	Status
	ft	%	%	%	%	
Plane 28	285.00	**0.79**	0.54	**0.84**	0.50	NG
Plane 27	275.00	0.42	0.30	0.45	0.50	OK
Plane 26	265.00	0.44	0.32	0.46	0.50	OK
Plane 25	255.00	0.45	0.35	0.47	0.50	OK
Plane 24	245.00	0.46	0.38	0.49	0.50	OK
Plane 23	235.00	0.48	0.42	0.50	0.50	OK
Plane 22	225.00	0.49	0.45	0.52	0.50	NG
Plane 21	215.00	0.50	0.48	0.54	0.50	NG
Plane 20	205.00	0.52	0.51	0.57	0.50	NG
Plane 19	195.00	0.53	0.54	0.61	0.50	NG
Plane 18	185.00	0.53	0.57	0.64	0.50	NG
Plane 17	175.00	0.54	0.59	0.66	0.50	NG
Plane 16	165.00	0.54	0.61	0.69	0.50	NG
Plane 15	155.00	0.55	0.63	0.71	0.50	NG
Plane 14	145.00	0.54	0.65	0.73	0.50	NG

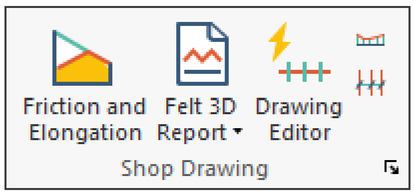
4.2.7 Drift Amplification Factor

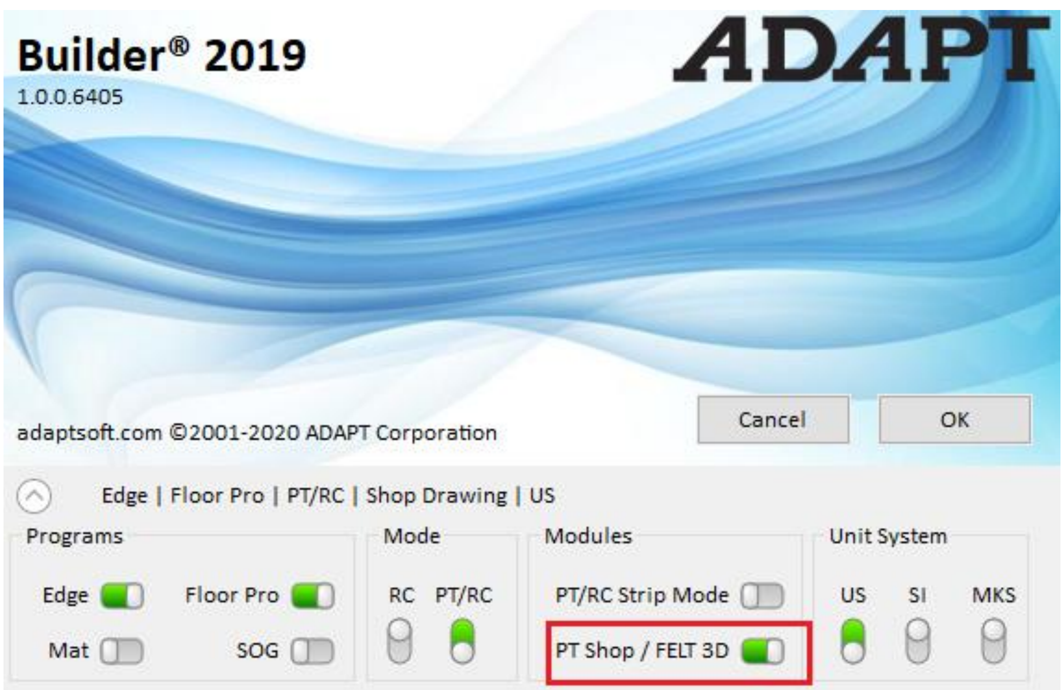
This sheet reports the directional drift amplification % for selected P-Delta combinations. Results are shown for each column at every plane in the structure and code checked against the user-defined maximum allowable. %. If any of the direction values (X, Y or combined) exceed the allowable the program will flag this row (indicating a single column for a single load combination result) as **NG**. If the values are less than allowable the row will be flagged as **OK**. The ratio can be taken as the 2nd order drift/1st order drift. The program calculates drift values at the top and bottom of each column. The program calculated drifts at both locations and uses the maximum ratio for this report.

Ref plane	Station	LC	Type	Drift X Amp	Drift Y Amp	Drift Combined Amp	Allowable	Status
				%	%	%	%	
Plane 28	Column 251	PDSX	P_Delta	1.04	0.96	1.03	1.40	OK
Plane 28	Column 251	PDSY	P_Delta	1.02	1.10	1.09	1.40	OK
Plane 28	Column 252	PDSX	P_Delta	1.04	0.99	1.04	1.40	OK
Plane 28	Column 252	PDSY	P_Delta	1.03	1.10	1.09	1.40	OK
Plane 28	Column 253	PDSX	P_Delta	1.05	0.99	1.05	1.40	OK
Plane 28	Column 253	PDSY	P_Delta	1.04	1.10	1.09	1.40	OK
Plane 28	Column 254	PDSX	P_Delta	1.05	0.99	1.05	1.40	OK
Plane 28	Column 254	PDSY	P_Delta	1.04	1.10	1.09	1.40	OK
Plane 28	Column 255	PDSX	P_Delta	1.05	1.03	1.05	1.40	OK
Plane 28	Column 255	PDSY	P_Delta	1.04	1.10	1.09	1.40	OK
Plane 28	Column 256	PDSX	P_Delta	1.03	1.04	1.04	1.40	OK
Plane 28	Column 256	PDSY	P_Delta	1.02	1.10	1.09	1.40	OK
Plane 28	Column 257	PDSX	P_Delta	1.05	1.05	1.05	1.40	OK
Plane 28	Column 257	PDSY	P_Delta	1.04	1.11	1.09	1.40	OK
Plane 28	Column 281	PDSX	P_Delta	1.05	1.05	1.05	1.40	OK

4.3 Felt 3D Reports

When the ‘PT Shop/FELT 3D’ module is selected in the program splash screen, additional features are added to the Tendon ribbon in the ‘Shop Drawing’ panel. These are shown below and were described in more detail in Section 2.5 of this document.





If modeled tendons have been set to use the Calculated Long-Term Loss option, the program will generate a compiled PDF report summarizing tendon loss calculations including graphical views of the tendons horizontal and vertical profile and the loss diagrams. The ‘Friction and Elongation’ calculation option should be processed before producing the report.

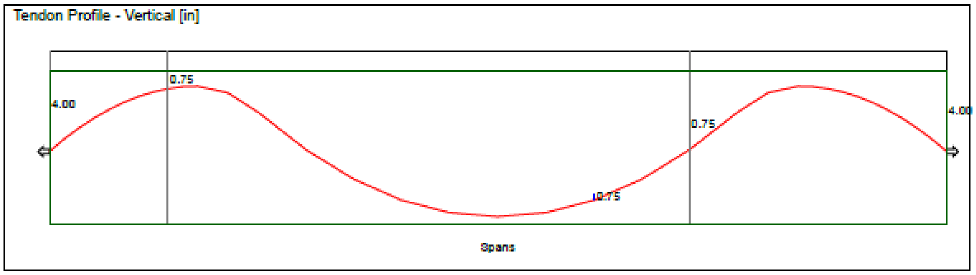
The report can be produced for a selected tendon/s or all tendons if none are selected. Depending on the number of tendons being processed, the report can take several minutes to produce and is done so as a .PDF document. The reports are saved in the model solution folder in a sub-folder called “FeltDesign.” Select the button called ‘FELT 3D Report’ from the panel shown above or use Reports→Tendon→PDF Reports→Felt 3D Report or FELT 3D-Single Strand Report. Note the pull-down arrow on the panel button allows the Single Strand option to be selected. When using the Single Strand option the program produces the report for a single tendon if the modeled tendon contains multiple strands. If this option is not used, the calculations will be based on the total of all strands for the modeled tendon. In this context a “strand” refers to a single tendon or cable. An example of a completed FELT 3D report is shown below.

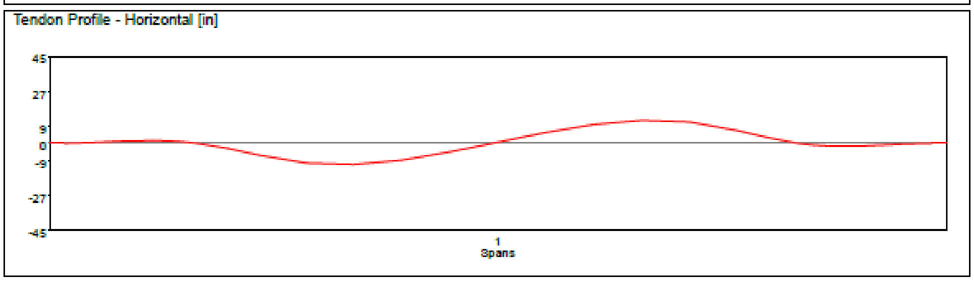
ADAPT-FELT 3D (Version: 2019)
POST-TENSIONING STRESS LOSS & ELONGATION CALCULATIONS

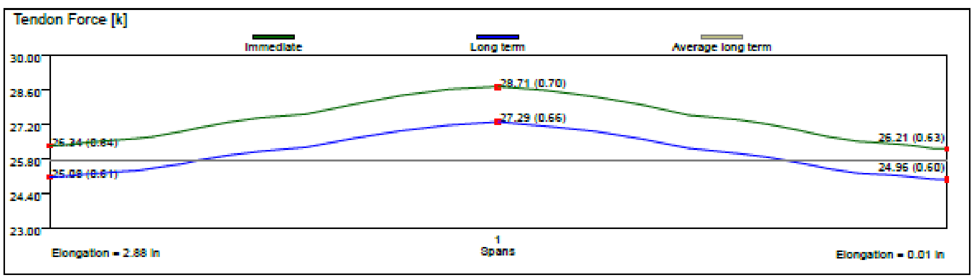
File Name: test.adm | Project Title: General name | Date: 11-05-19 Time: 11:50
Tendon Label, ID: "Tendon 14", 14 | Specific Title: Specific name

Fu = 270.00 ksi	System = Unbonded	Type of strand = Low Lax	Humidity = 80.00 %
Es = 28000.00 ksi	K = 0.0010 rad/ft	Age@stressing = 5.00 days	V/S = 4.00 in
Astrand = 0.153 in2	Mu = 0.07/radian	Ec @stressing = 1500.00 ksi	fci = 0.15 ksi
Num of Strands = 1.00	L = 36.42 ft	Ec @28 days = 3600.00 ksi	Kcr = 1.60

Stressing ends: Live / 0.80 / 0.25 in | Live / 0.80 / 0.25 in







Average values (after release/final)
Force = 27.07 / 25.76 k
Stress = 176.94 / 168.34 ksi

Long-term loses (Calculated)
Total Losses = 8.59 ksi
Shortening = 1.40 ksi | Creep = 1.87 ksi
Shrinkage = 2.79 ksi | Relaxation = 2.53 ksi

Jacking Force / Stress
Left pull = 33.05 k / 216.00 ksi
Right pull = 33.05 k / 216.00 ksi

Anchor set influence (corresponding stress/ratio)
Left pull = 18.18 ft (188 ksi / 0.70)
Right pull = 20.93 ft (188 ksi / 0.70)

Elongation (before/after anchor set)
Left pull = 3.13 in / 2.88 in
Right pull = 0.26 in / 0.01 in
Total after anchor set = 2.89 in
Total after anchor set/L = 0.08 in/ft

Critical Stress Ratios (at stressing/at anchorage)
Left pull = 0.80 / 0.64
Right pull = 0.80 / 0.63
Maximum along tendon = 0.70
Minimum along tendon = 0.63