PTKalc

PTKalc User, Analysis and Design Manual

Document Revision

Revision Title	Rev #	Issue Date	By
First Issue	0	10th August 2026	RK

Table of Contents

1. Introduction

1.1. Overview

The structural behaviour of post-tensioned concrete members is governed not only by externally applied loading but also by the internal forces introduced through prestressing tendons. Unlike conventional reinforced concrete, where all structural actions originate from external loads, prestressed members contain an initial system of internal forces that modifies both the stress distribution within the concrete and the overall response of the structure.

This approach allows prestressing to be incorporated into the structural analysis using the same finite element solver employed for conventional dead, live and environmental loading. Once the equivalent prestress loading has been assembled, the global structural response is obtained through a single unified analysis procedure. The resulting bending moments, shear forces, reactions and deflections therefore include the effects of prestress in a manner fully compatible with all other applied loading.

The equivalent load method is a well-established engineering approach for representing the global structural effects of prestressing tendons and is widely used in the analysis of post-tensioned concrete structures. PTKalc adopts the same underlying engineering principles while implementing its own numerical procedures for profile generation, load conversion and finite element assembly.

1.2. Scope and Source Basis

This document describes the principal analysis and design methodologies implemented within PTKalc, including structural modelling, tendon-profile generation, equivalent prestress loading, prestress losses, finite-element analysis, flexural design, serviceability assessment and long-term deflection. The document is intended to provide a transparent engineering description of the computational procedures used by the software.

1.3. Philosophy of the Equivalent Load Method

Prestressing tendons do not normally apply their force directly to the surrounding concrete along their entire length. Instead, the tendon is restrained only where its direction changes or where it is anchored.

Consider a perfectly straight tendon placed concentrically within a beam. Although a substantial tensile force exists within the tendon, no transverse loading is applied to the concrete member. The tendon simply introduces an axial compressive force throughout the section.

When the tendon follows a curved profile, however, equilibrium requires the concrete to provide a continuous lateral reaction to restrain the change in tendon direction. By Newton's Third Law, the tendon therefore applies an equal and opposite transverse force to the concrete. This distributed reaction is precisely equivalent to an external load acting upon the structure.

The magnitude of this equivalent loading depends entirely upon the curvature of the tendon profile.

A straight tendon produces no equivalent transverse loading.

A tendon with constant curvature produces a constant distributed load.

Abrupt changes in tendon slope generate concentrated forces at the points of deviation.

Consequently, the structural effects of prestressing may be represented entirely by replacing the tendon with an equivalent system of externally applied loads. Once this transformation has been completed, the global structural effects of the tendon may be analysed using the same finite-element procedures used for conventionally applied loading.

This observation forms the foundation of the equivalent load method and allows prestressing to be analysed using standard finite element procedures without requiring specialised prestressed beam elements.

1.4. Prestress Actions Considered by PTKalc

PTKalc considers the actions generated by prestressing at two distinct levels.

The first level consists of global structural actions. At this stage, tendon profiles are converted into equivalent distributed and concentrated loads, which are assembled into the global load vector and analysed together with all externally applied loading. The resulting structural analysis determines the bending moments, shear forces, support reactions and member deflections throughout the complete structure.

The second level consists of local cross-sectional actions. Following completion of the global analysis, the effective prestressing force, together with its eccentricity relative to the section centroid, is used in detailed cross-sectional calculations. These calculations determine concrete stresses, steel stresses, cracking behaviour, ultimate flexural strength, shear capacity and effective stiffness for deflection assessment.

This separation between global structural analysis and local section design provides a clear computational framework while ensuring that both aspects of prestress behaviour are consistently represented.

1.5. Advantages of the Method

The equivalent load methodology adopted within PTKalc offers several important engineering advantages.

- The method is directly compatible with conventional finite element beam analysis and therefore requires no specialised prestressed beam elements.
- Multiple tendon profiles may be analysed simultaneously regardless of their geometry, tendon force or termination location.
- Continuous tendons extending over several spans may be analysed without modification of the underlying finite element formulation.
- Partial tendons terminating within individual spans are accommodated naturally through the equivalent load generation process.
- Since prestress is represented by external loading, all standard structural analysis procedures—including load combinations, cracking analysis, long-term deflection calculations and reinforcement design—operate using a common structural model.

2. User Interface

2.1. Setting up a new beam

Below screenshot is the main user interface of PTKalc, along with annotations for users to start a new problem. The user interface is fairly simple.

Main steps include:

1- Defining the materials and criteria – see Figure 3 (including concrete strength, modulus of rupture, creep factor, concrete covers, PT tendons and environmental conditions)
2- Define geometry – see Figure 1 (by inserting number of spans, span lengths, supporting conditions including columns over and under, plus drapes if beam is PT)
3- Insert loads – see Figure 4 (user may select load case, and spans, then insert loads, being uniformly distributed loads or point loads). Note the programme currently allows for most typical loading scenarios, trapezoidal loads are not available. User may apply equivalent uniform loads in case of band beams supporting different tributary areas
4- Review load combinations – see Figure 5 (note the program is provided with default load cases and combinations – user is able to edit these and add / delete existing load combinations)
5-

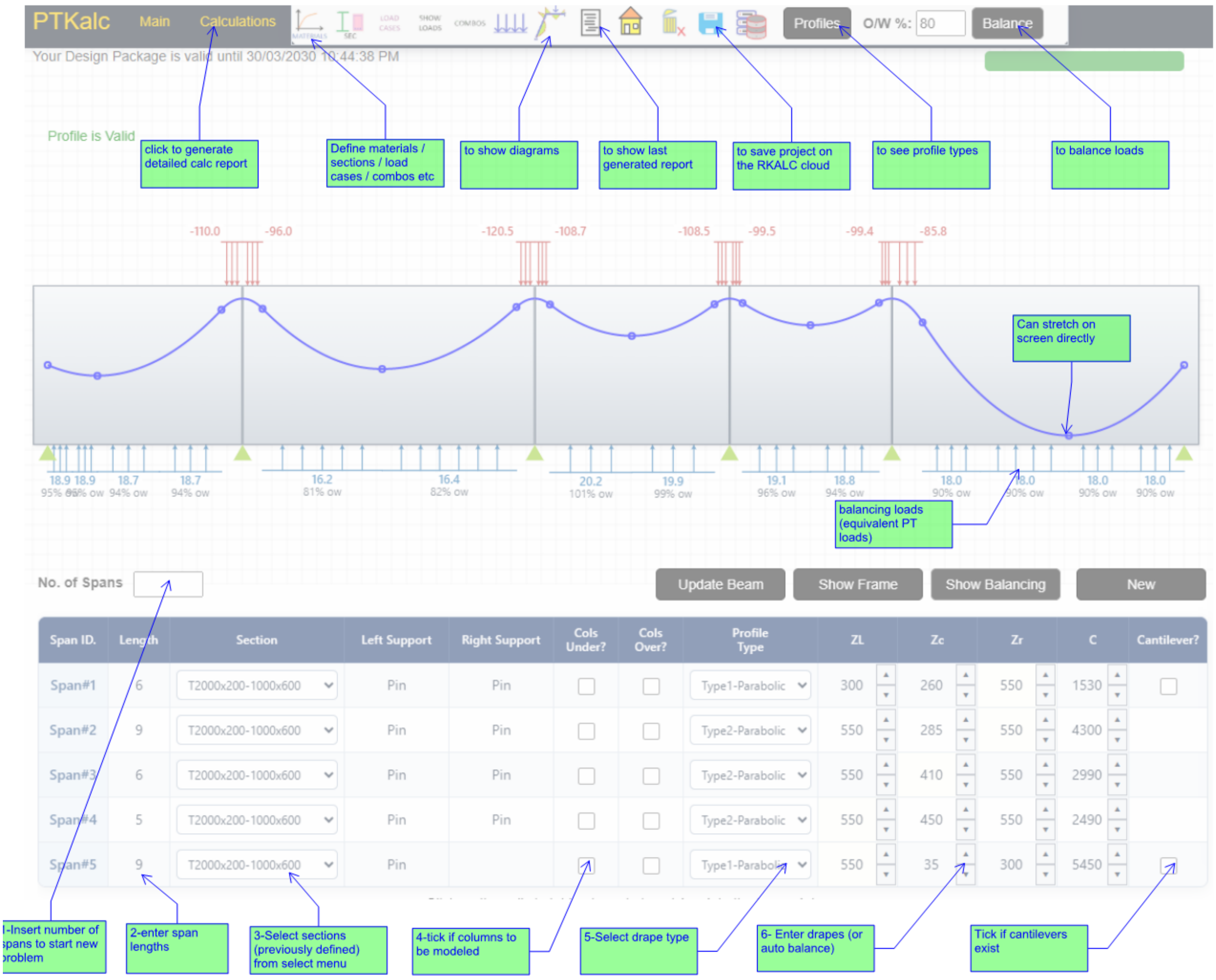


Figure 1-User Interface

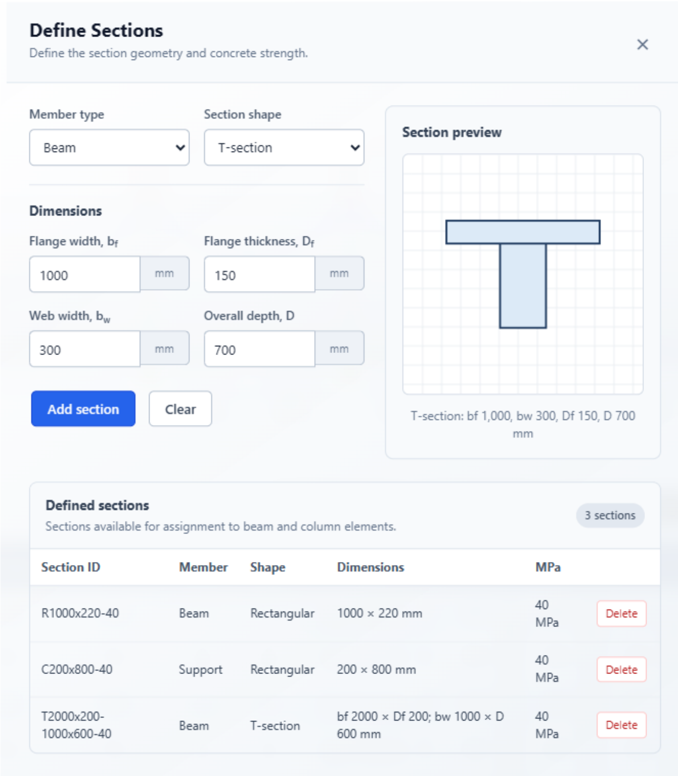
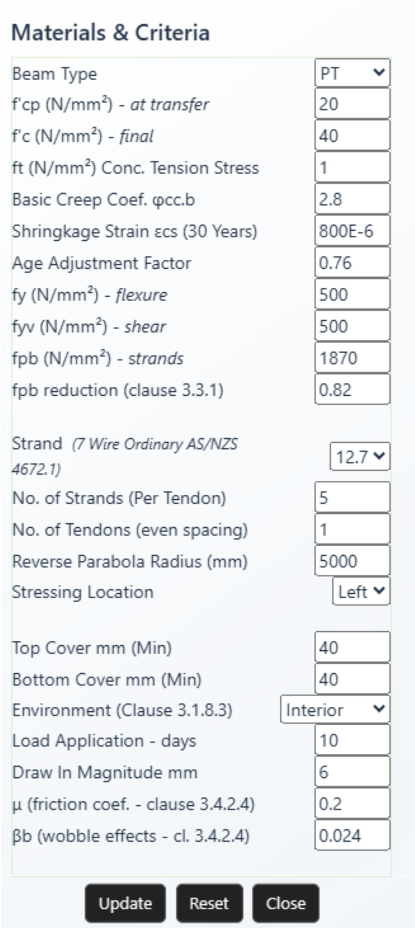
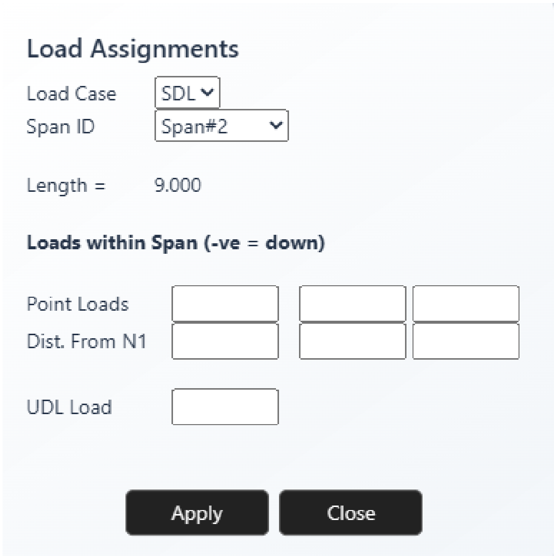


Figure 2-Section Definition





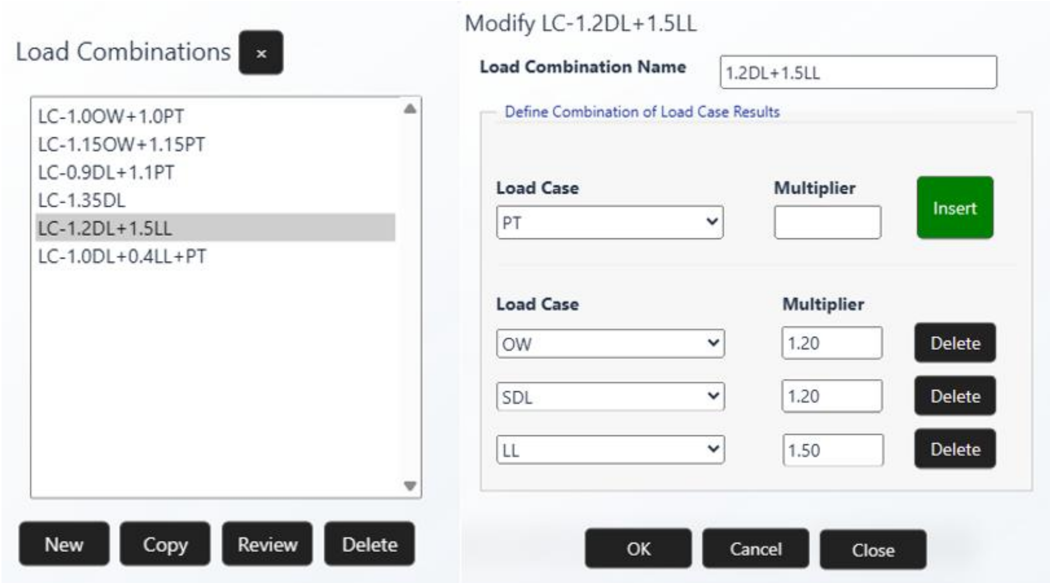


Figure 5-Load Combinations

2.2. Reviewing Results

Results can be reviewed using the Diagrams dialog box show in Figure 6

1- Reactions, for selected load case
2- Bending moments, shears for selected load case
3- Deformations for selected load case. Note that raw load cases will show elastic deformations only, whereas load combinations will show both of elastic and long term deformations resulting from creep, shrinkage and cracked sections
4- Elastic Stress block for selected load case
5- Flexural and shear reinforcement results for selected combo
6- Flexural and shear reinforcement for Envelope of all cases (basically the design)

For detailed results, click on Calculations to generate a full report

2.3. Saving and opening a saved project

Click on save menu to save a project, a dialog box appears to insert project name and model reference. The project will be saved in RKALC cloud, this will be linked to user account (email address through which user is logging in to RKALC)

If a project is saved, click on database menu to open the project.

Figure 7 – Showing moment diagrams

Figure 8 – showing deformations (elastic and long term)

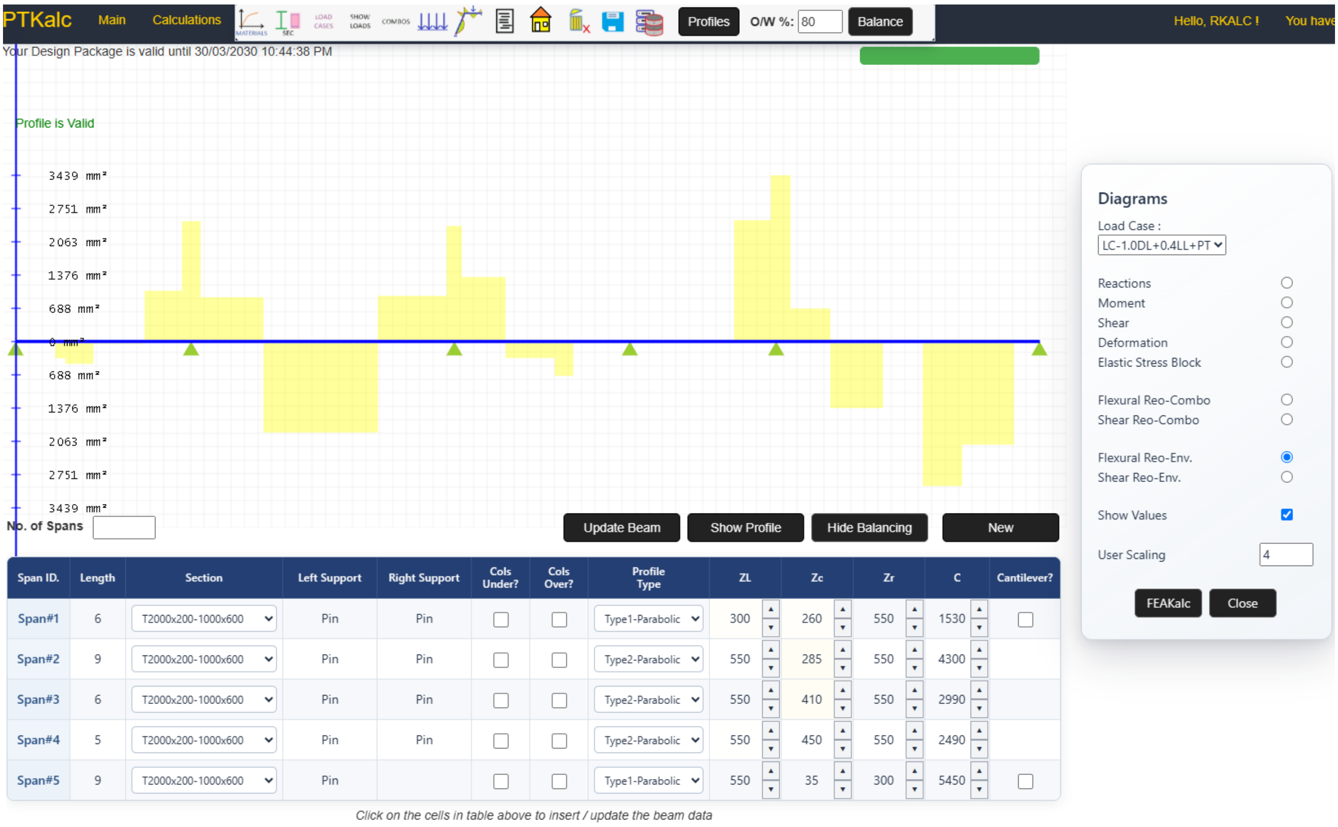


Figure 9 – Reinforcing Envelope (flexure)

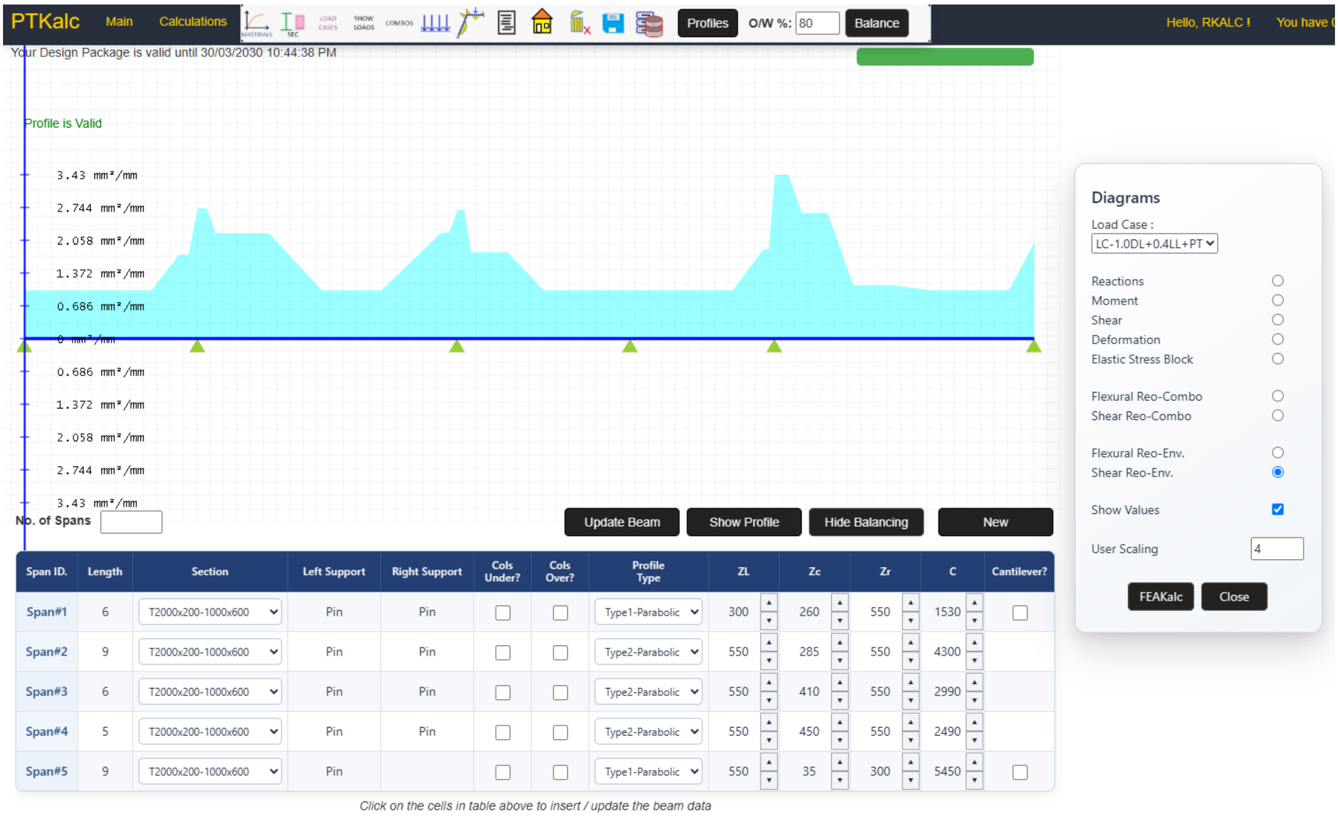


Figure 10 - Reinforcing Envelope (Shear)

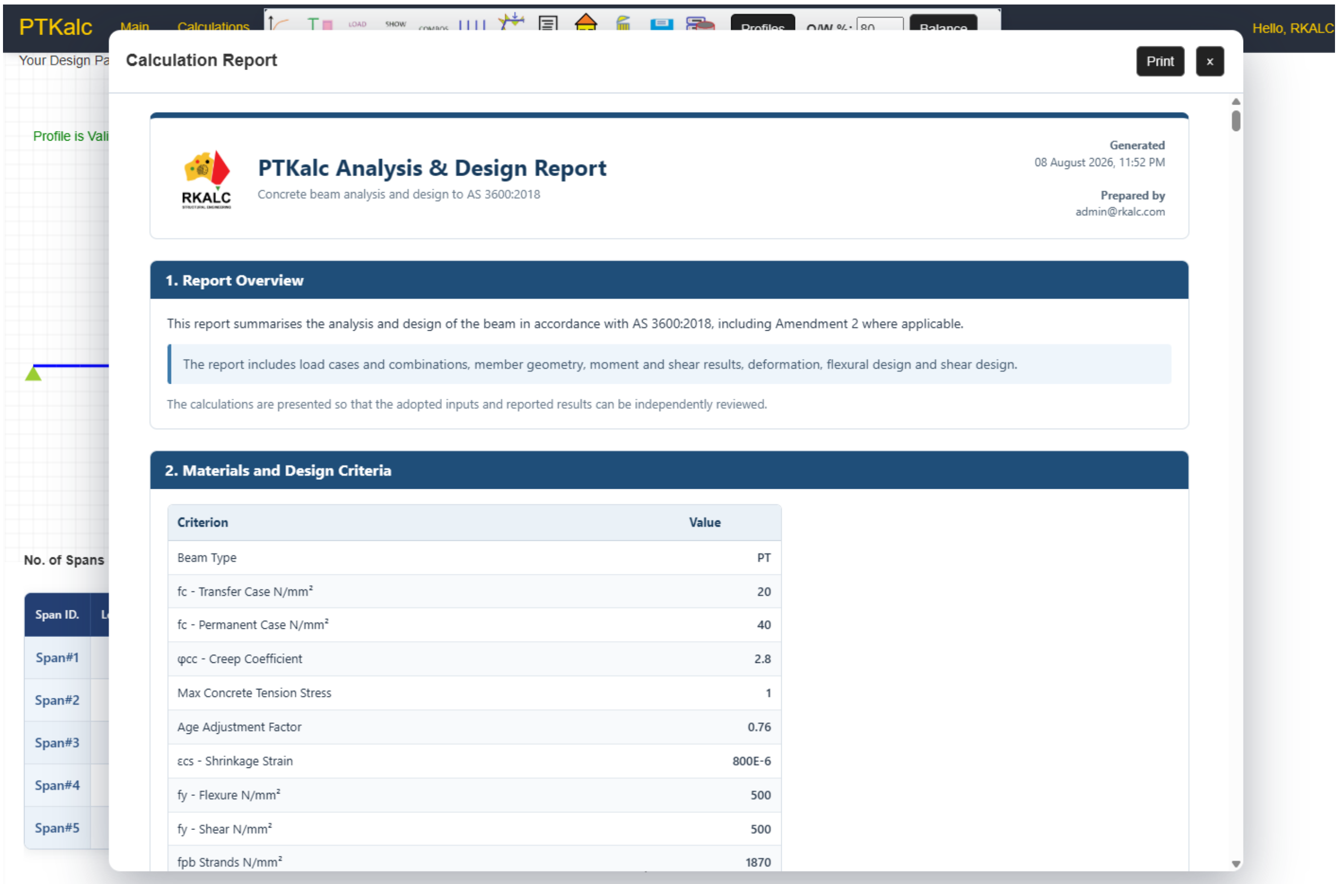


Figure 11 – PTKalc Calculation Report

2.4. Reviewing Deformation

User may review several scenarios for deflection to establish upper and lower bounds, for example:

1- Adjust modulus of rupture for concrete
2- Adjust shrinkage and creep factors
3- Adjust covers, age factor and so on

Refer to section 14- Serviceability and Deflection Analysis and verification documentation

3. Mathematical Representation of Tendon Profiles

3.1. Introduction

The structural actions generated by a post-tensioning tendon are governed entirely by its geometry. While the magnitude of the prestressing force determines the intensity of the actions imposed upon the structure, it is the tendon profile that controls where these actions occur and how they are distributed throughout the member.

Within PTKalc, each tendon is represented by a continuous mathematical function describing its vertical position relative to a fixed reference axis. This function provides the basis for calculating tendon eccentricity, slope and curvature, from which the equivalent structural loading is subsequently derived.

Unlike traditional manual calculations, which frequently assume a single idealised parabola over an entire span, PTKalc represents tendon geometry numerically, allowing complex tendon layouts to be analysed without requiring separate analytical solutions for each configuration. Straight segments, parabolic drapes, reverse curves, continuous multi-span tendons and partially terminated tendons are all treated using the same general mathematical framework.

The objective of this chapter is therefore to establish the mathematical description of tendon geometry before considering the forces generated by the tendon in the following chapter.

3.2. Coordinate System

A consistent coordinate system is required in order to describe the tendon position throughout the structure.

The longitudinal axis of the member is defined by the global coordinate

$$x$$

measured along the centroidal axis of the beam from the origin of the finite element model.

The tendon profile is represented by the vertical coordinate

$$y(x)$$

measured positive upwards from the chosen datum.

For structural calculations, however, absolute elevation is of secondary importance. The critical parameter governing prestress behaviour is the tendon eccentricity relative to the centroid of the concrete section.

Accordingly,

$$e(x) = y(x) - y_c(x)$$

where

- $e(x)$ is tendon eccentricity,
- $y(x)$ is tendon elevation,
- $y_c(x)$ is the centroidal elevation of the concrete section.

For prismatic beams having constant depth,

$$y_c = \text{constant}$$

and therefore

$$e(x) = y(x) - C$$

where C is the constant centroidal elevation.

When adjacent spans have different depths, PTKalc adjusts the soffit-based tendon offset to preserve the physical tendon elevation across the depth change. Section centroidal properties are introduced subsequently for cross-sectional stress and eccentricity calculations.

Accordingly, tendon geometry and equivalent loading are generated from section-depth-based profile offsets, while local section stresses use the calculated centroid, area, inertia and top and bottom fibre distances.

3.3. Description of Tendon Profiles

The tendon profile defines the path followed by the prestressing strand through the concrete member.

Several profile types are commonly employed in practical design, each producing different structural effects.

Straight Tendons

The simplest profile consists of a tendon having constant eccentricity throughout its length.

Mathematically,

$$e(x) = e_0$$

where

$$e_0 = \text{constant}$$

Since

$$\frac{de}{dx} = 0$$

and

$$\frac{d^2e}{dx^2} = 0$$

the tendon possesses neither slope variation nor curvature.

Consequently, a straight tendon introduces only axial compression and an eccentric prestress moment

$$M_p = Pe_0$$

without generating any equivalent transverse loading along the beam.

Parabolic Profiles

The majority of bonded and unbonded post-tensioned beams employ parabolic tendon profiles.

These profiles provide smooth curvature while maintaining simple construction requirements.

The general equation is

$$e(x) = ax^2 + bx + c$$

where the constants

$$a, \ b, \ c$$

are determined from the specified tendon elevations at the supports and the low point within the span.

Parabolic profiles possess constant curvature,

$$\frac{d^2e}{dx^2} = 2a$$

which results in a uniform equivalent distributed load.

This property explains why parabolic tendons are particularly efficient for balancing uniformly distributed dead loading.

Harped Profiles

Where tendon geometry changes abruptly at deviator points, the tendon consists of straight segments connected by discrete changes in direction.

The profile may be written as

$$e(x) = \begin{cases} m_1x + c_1, & x < x_d \\ m_2x + c_2, & x > x_d \end{cases}$$

where

$$x_d$$

defines the deviator location.

Within each straight segment,

$$\frac{d^2e}{dx^2} = 0$$

so no distributed load exists.

However, the sudden change in slope generates a concentrated force at the deviator, the derivation of which is presented in Chapter 3.

Multi-span Profiles

Continuous structures require tendon profiles extending over several spans.

Rather than treating each span independently, PTKalc models the tendon as a continuous profile satisfying geometric compatibility at every support.

Accordingly,

$$e_i(L_i) = e_{i+1}(0)$$

ensuring continuity of elevation between adjacent spans.

Where smooth tendon curvature is required,

$$\frac{de_i}{dx} = \frac{de_{i+1}}{dx}$$

may also be enforced.

This continuous representation eliminates artificial discontinuities in the equivalent loading and produces a more realistic simulation of tendon behaviour.

3.4. Profile Continuity

The mechanical behaviour of prestressing tendons depends not only upon their position but also upon the continuity of the profile.

A physically realistic tendon cannot contain instantaneous jumps in elevation.

Therefore,

$$e(x)$$

must always remain continuous.

In addition, tendon construction generally requires smooth transitions between adjacent profile segments.

This requires continuity of the first derivative,

$$\frac{de}{dx}$$

except where intentional deviators are introduced.

PTKalc therefore distinguishes between two forms of profile transition.

The first consists of smooth geometric transitions in which both elevation and slope remain continuous.

The second consists of intentional changes in tendon direction, where elevation remains continuous but slope changes abruptly.

These discrete slope changes give rise to concentrated prestress forces, which form an important component of the equivalent loading system.

3.5. Slope of the Tendon

The slope of the tendon describes the inclination of the prestressing force relative to the beam axis.

Differentiating the eccentricity function gives

$$\theta(x) = \frac{de(x)}{dx}$$

where

$$\theta$$

is the tendon inclination.

For a parabolic profile,

$$e(x) = ax^2 + bx + c$$

therefore

$$\theta(x) = 2ax + b$$

showing that the tendon inclination varies linearly along the span.

The significance of tendon slope extends beyond simple geometry.

Changes in slope produce vertical components of prestressing force, while changes in slope between adjacent sections ultimately generate the distributed loading applied to the structure.

3.6. Curvature of the Tendon

The quantity governing equivalent prestress loading is not the tendon elevation itself, nor its slope, but its curvature.

Differentiating the slope gives

$$\kappa(x) = \frac{d^2 e(x)}{dx^2}$$

where

$$\kappa$$

is the geometric curvature of the tendon profile.

For the general parabolic profile,

$$e(x) = ax^2 + bx + c$$

the curvature becomes

$$\kappa = 2a$$

which is constant throughout the span.

This remarkably simple result forms the basis of the classical equivalent load method.

Since the prestressing force remains essentially constant along the tendon between anchorages, a constant curvature produces a constant transverse reaction between the tendon and the surrounding concrete. Consequently, a parabolic tendon generates a uniformly distributed equivalent load, making it ideally suited to balancing uniformly distributed gravity loading.

The relationship between tendon curvature and equivalent structural loading is derived rigorously in the following chapter, where the governing equations are developed directly from the equilibrium of an infinitesimal tendon element.

4. PTKalc Tendon Profile Types and Equivalent Prestress Loading

4.1. Introduction

PTKalc does not use a single generic tendon shape for every span. The prestressing engine distinguishes between several profile families according to the structural position of the span and the intended form of equivalent loading.

The profile classification used in the current PTKalc implementation is:

- Type 1 – end-span parabolic profile;
- Type 2 – internal-span parabolic profile;
- Type 3 – cantilever parabolic profile;
- Type 4 – simply supported parabolic profile;
- Type 5 – two-segment harped profile producing a concentrated deviation load;
- Type 6 – single inclined straight profile used at an end or cantilever-oriented span, producing an end transverse force component;

Separate left- and right-hand procedures are used where the tendon geometry is not symmetric. This applies particularly to end spans and cantilevers, where the stressing direction, adjacent support condition and reverse-curvature region affect the adopted segment geometry.

The calculations are organised around two linked operations:

1. construction of the tendon geometry; and
2. conversion of that geometry into equivalent finite element loading.

The drawing procedures are therefore not merely graphical functions. They establish the individual profile segments, determine the slope changes used in the friction calculation, calculate the equivalent prestress forces, and pass the resulting loads to the finite element model.

4.2. General Profile Parameters

The profile procedures use a common set of geometric quantities.

For a span of length

$$L$$

the tendon elevations are generally represented by:

$$Y_a$$

at the left end,

$$Y_b$$

at the right end, and

$$Y_c$$

at an internal high or low point located at distance

$$C$$

from the left end.

The exact interpretation depends upon the profile type and beam orientation, but the general arrangement is:

$$(0, Y_a)$$

at the left boundary,

$$(C, Y_c)$$

at the principal internal profile point, and

$$(L, Y_b)$$

at the right boundary.

The code converts these elevations from a soffit-based coordinate to an internal drawing coordinate by subtracting the unscaled beam depth:

$$\begin{gathered} Y_a^* = -D + Y_a \\ Y_b^* = -D + Y_b \\ Y_c^* = -D + Y_c \end{gathered}$$

where

$$D$$

is the unscaled beam depth.

This transformation is used for profile generation and tendon-table output. For equivalent-load calculations, the original tendon offsets are restored by adding the unscaled depth before the load-calculation procedures are called.

This separation is important. The graphical coordinate system is not itself the engineering eccentricity system. PTKalc converts between the display geometry and the physical tendon offsets used in the prestress calculations.

4.3. Piecewise Profile Representation

The parabolic profiles are represented as a sequence of individual quadratic segments rather than as one polynomial extending over the entire span.

A general quadratic segment is written as

$$y(x) = A(x - x_0)^2 + y_0$$

with slope

$$\frac{dy}{dx} = 2A(x - x_0)$$

and constant second derivative

$$\frac{d^2y}{dx^2} = 2A.$$

Where a segment terminates with zero slope, the quadratic origin is placed at the tangent point. This allows PTKalc to construct profiles that are horizontal at low points or high points and smoothly approach support regions.

Some profile types contain up to four geometric regions:

Segment 1 → Segment 2 → Segment 3 → Segment 4.

The transition between adjacent parabolas is obtained either analytically or numerically.

For some left-side transitions, the switch location is obtained from a discriminant calculation. The code forms

$$AA = \frac{C^2}{Y_a - Y_c}$$
$$BB = 4RC$$
$$CC = 4R^2(Y_a - Y_c)$$

and evaluates

$$\Delta = BB^2 - 4AA\,CC.$$

Where

$$\Delta \geq 0,$$

the transition roots are

$$x_1 = \frac{-BB + \sqrt{\Delta}}{2AA}$$

and

$$x_2 = \frac{-BB - \sqrt{\Delta}}{2AA}.$$

The applicable transition distance, denoted in the implementation as X_{12}, is selected from the physically relevant root.

For other profile regions, PTKalc evaluates two adjacent parabola equations along the span and identifies the location at which their elevations coincide:

$$|y_3(x) - y_4(x)| < \varepsilon$$

where the numerical tolerance adopted in the current implementation is

$$\varepsilon = 0.0001.$$

The identified location is stored as the profile-switch coordinate.

This produces a piecewise tendon profile in which each component parabola represents a distinct curvature region and therefore a distinct equivalent distributed load.

4.4. Fundamental Equivalent-Load Relationship

For a tendon carrying effective force

$$P$$

and following a vertical profile

$$y(x),$$

the vertical equivalent loading is related to tendon curvature by

$$w(x) = P\frac{d^2y}{dx^2}.$$

The algebraic sign depends upon the coordinate and load conventions adopted in the analysis model. Within PTKalc, the calculated load values are subsequently passed to the finite element table with explicit positive or negative signs for each profile region.

For a parabolic segment

$$y(x) = A(x - x_0)^2 + y_0,$$

the second derivative is

$$\frac{d^2y}{dx^2} = 2A.$$

Therefore,

$$w = 2PA.$$

If a parabola has total vertical rise or fall

$$a$$

over horizontal projection

$$l,$$

and is tangent at one end, its coefficient is

$$A = \frac{a}{l^2}.$$

The equivalent load is therefore

$$\boxed{w = \frac{2Pa}{l^2}}$$

which is the governing expression used throughout the PTKalc profile-load routines.

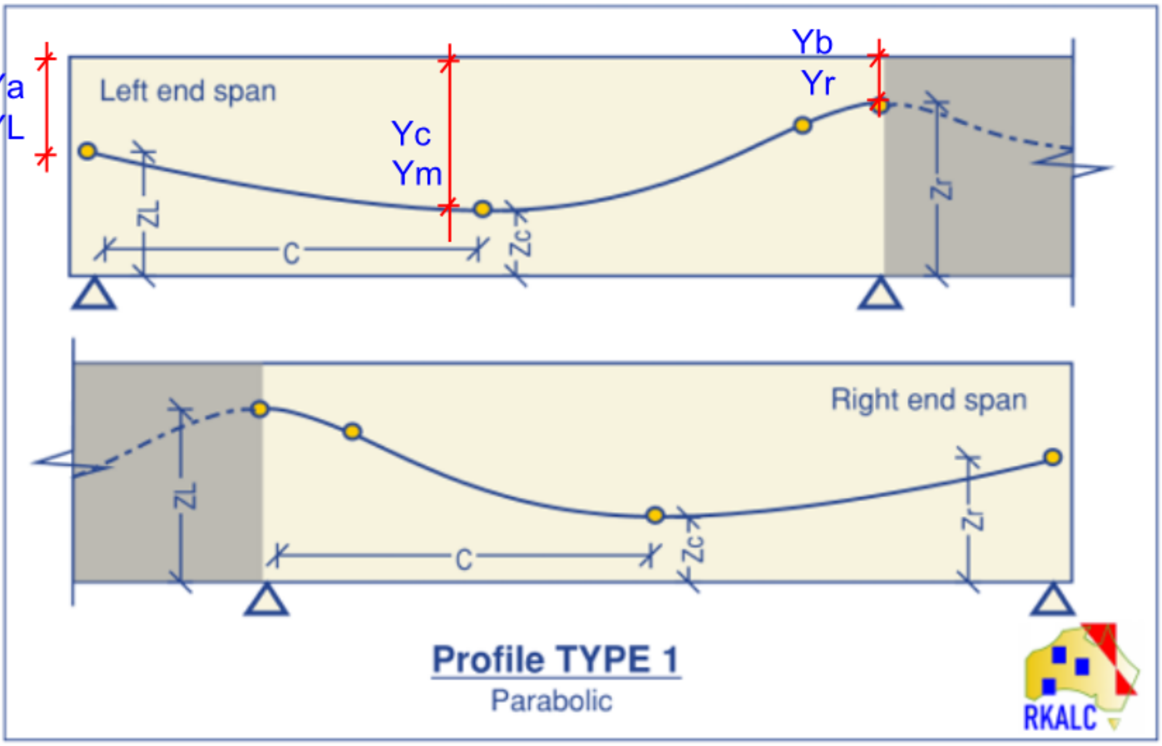
The code repeatedly applies this relationship in the form

$$W_i = \frac{2Fa_i}{l_i^2},$$

where:

- F is the effective tendon force after friction adjustment;
- a_i is the vertical component assigned to profile segment i;
- l_i is the horizontal length of that segment;
- W_i is the uniform equivalent load applied over the segment.

4.5. Type 1 Profile – End-Span Parabolic Tendon



General arrangement

A Type 1 profile represents a parabolic tendon in an end span.

Two variants are implemented:

- Type 1 left;
- Type 1 right.

The two variants are not simple mirror-image drawing operations. They use different profile-segment arrangements because one end of the tendon is associated with the exterior boundary while the other connects to an internal support region.

The profile may contain up to four equivalent-load zones:

$$W_1, \quad W_2, \quad W_3, \quad W_4.$$

One exterior zone has zero loading because the corresponding end of the profile has no reverse-curvature segment.

Type 1 left geometry

For the left-oriented end-span profile, the first active segment extends from

$$x = \mathbf{0}$$

to

$$x = \boldsymbol{C}.$$

Its equation is

$$y_2(x) = \frac{(x-C)^2}{C^2/(Y_a - Y_c)} + Y_c.$$

This can be rewritten as

$$y_2(x) = \frac{Y_a - Y_c}{C^2}(x-C)^2 + Y_c.$$

The corresponding slope is

$$\frac{dy_2}{dx} = 2\frac{Y_a - Y_c}{C^2}(x-C).$$

The segment approaches the point

$$(C, Y_c)$$

with zero slope.

The right-hand portion of the span is constructed from two parabolic regions. The first is

$$y_3(x) = A_3(x-C)^2 + Y_c$$

where

$$A_3 = \frac{1}{-2R + (L-C)^2/(Y_b - Y_c)}.$$

The terminal reverse-curvature segment is

$$y_4(x) = A_4(L-x)^2 + B_4 + Y_c$$

where

$$A_4 = \frac{1}{-2R}$$

and

$$B_4 = Y_b - Y_c.$$

The transition coordinate x_s is the point at which

$$y_3(x_s) = y_4(x_s).$$

Type 1 left equivalent loads

For the Type 1 left calculation, PTKalc defines

$B = L - x_s, C = C,$

and

$$D = L - C.$$

The total left-side drape is

$$a_2 = Y_m - Y_l.$$

The right-side drape is divided between two curvature regions:

$$a_4 = \frac{B}{D}(Y_m - Y_r)$$

and

$$a_3 = Y_m - Y_r - a_4.$$

The resulting equivalent loads are

$$W_1 = 0,$$
$$\boxed{W_2 = \frac{2Fa_2}{C^2}}$$
$$\boxed{W_3 = \frac{2Fa_3}{(D-B)^2}}$$

and

$$\boxed{W_4 = \frac{2Fa_4}{B^2}.}$$

The zero value of W_1 reflects the absence of an exterior reverse-curvature segment on the left side.

In the finite element model, the W_2 and W_3 regions are subdivided into shorter elements for greater load-placement accuracy. The terminal W_4 region is entered as a separate element load.

Type 1 right geometry

The right-oriented Type 1 profile begins with a reverse-curvature region between

$$x = 0$$

and

$$x = X_{12}.$$

The first parabola is

$$y_1(x) = -\frac{x^2}{2R} + Y_a.$$

Its slope is

$$\frac{dy_1}{dx} = -\frac{x}{R}.$$

The second parabola extends between

$$X_{12}$$

and

$$C$$

and is given by

$$y_2(x) = \frac{(x-C)^2}{-2R + C^2/(Y_a - Y_c)} + Y_c.$$

The remaining right-hand segment extends from

$$C$$

towards

$$L$$

using a parabola centred at the internal profile point.

Type 1 right equivalent loads

The first vertical drape component is

$$a_1 = \frac{A}{C}(Y_m - Y_l)$$

where

$$A = X_{12}.$$

The remaining left-side drape is

$$a_2 = Y_m - Y_l - a_1.$$

The right-side drape is

$$a_3 = Y_m - Y_r.$$

The equivalent loads become

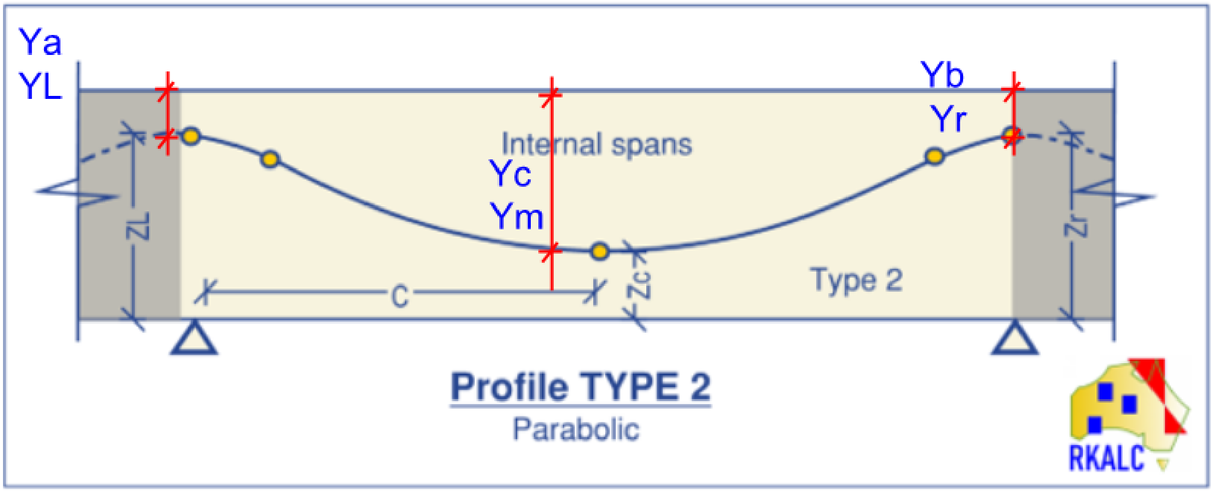
$$\boxed{W_1 = \frac{2Fa_1}{A^2}}$$
$$\boxed{W_2 = \frac{2Fa_2}{(C - A)^2}}$$
$$\boxed{W_3 = \frac{2Fa_3}{D^2}}$$

and

$$W_4 = 0.$$

The zero value of W_4 reflects the exterior boundary condition at the right-hand end.

4.6. Type 2 Profile – Internal-Span Parabolic Tendon



General arrangement

A Type 2 profile represents an internal span in a continuous beam.

Unlike an end span, an internal span may contain reverse-curvature regions at both ends. The profile therefore uses all four distributed-load components:

$$W_1, \quad W_2, \quad W_3, \quad W_4.$$

The geometry is formed from:

1. a left reverse-curvature parabola;
2. a parabola approaching the internal low or high point;
3. a parabola departing from that point;
4. a right reverse-curvature parabola.

Left-side geometry

The first segment is

$$y_1(x) = -\frac{x^2}{2R} + Y_a$$

for

$$0 \leq x \leq X_{12}.$$

The following segment is

$$y_2(x) = \frac{(x - C)^2}{-2R + C^2/(Y_a - Y_c)} + Y_c$$

for

$$X_{12} \leq x \leq C.$$

Right-side geometry

The first right-side parabola is

Note y values are taken from top of the beam (Y = Beam Depth – Z values)

$$y_3(x) = A_3(x - C)^2 + Y_c$$

where

$$A_3 = \frac{1}{-2R + (L - C)^2/(Y_b - Y_c)}.$$

The terminal parabola is

$$y_4(x) = -\frac{(L - x)^2}{2R} + (Y_b - Y_c) + Y_c.$$

The transition position x_s is determined by the intersection condition

$$y_3(x_s) = y_4(x_s).$$

The terminal transition length is

$$B = L - x_s.$$

Type 2 equivalent loads

The total left-side drape is separated into

$$a_1 = \frac{A}{C}(Y_m - Y_l)$$

and

$$a_2 = Y_m - Y_l - a_1.$$

Similarly, the total right-side drape is separated into

$$a_4 = \frac{B}{D}(Y_m - Y_r)$$

and

$$a_3 = Y_m - Y_r - a_4.$$

The equivalent distributed loads are

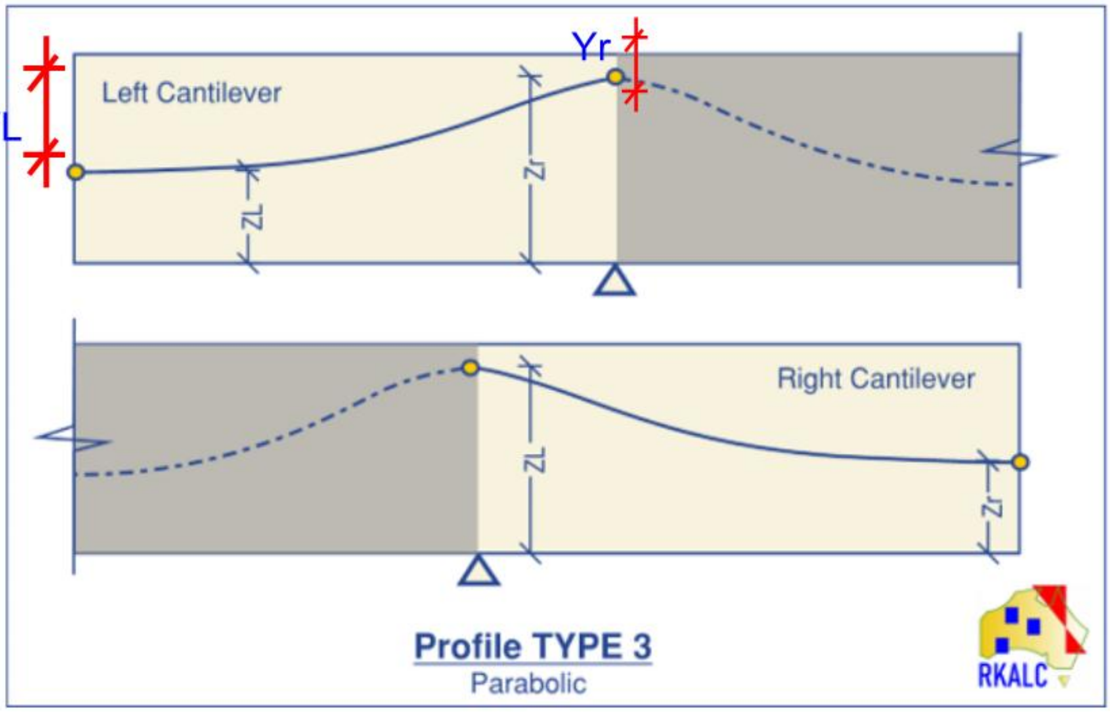
$$\boxed{W_1 = \frac{2Fa_1}{A^2}}$$
$$\boxed{W_2 = \frac{2Fa_2}{(C-A)^2}}$$
$$\boxed{W_3 = \frac{2Fa_3}{(D-B)^2}}$$

and

$$\boxed{W_4 = \frac{2Fa_4}{B^2}.}$$

This is the most general of the parabolic span-load formulations used in PTKalc. The Type 1 end-span formulas are effectively reduced forms of this internal-span arrangement, with one exterior reverse-curvature load set to zero.

4.7. Type 3 Profile – Cantilever Parabolic Tendon



General arrangement

A Type 3 profile represents a cantilever tendon.

Separate left and right procedures are used because the fixed support and free end are reversed.

Only two distributed-load components are required:

$$W_1$$

and

$$W_2.$$

These represent the two parabolic regions forming the cantilever tendon.

Type 3 left profile

For the left-oriented cantilever, the first parabola is

$$y_3(x) = A_3 x^2 + Y_c$$

where

$$A_3 = \frac{1}{-2R + L^2/(Y_b - Y_c)}.$$

The terminal reverse-curvature parabola is

$$y_4(x) = -\frac{(L-x)^2}{2R} + (Y_b - Y_c) + Y_c.$$

The switch point x_s is obtained from

$$y_3(x_s) = y_4(x_s).$$

The terminal length is then

$$B = L - x_s.$$

The total vertical difference is divided according to horizontal segment length:

$$a_R = (Y_l - Y_r)\frac{B}{L}$$

and

$$a_M = (Y_l - Y_r)\frac{L - B}{L}.$$

The equivalent loads are

$$\boxed{W_1 = \frac{2Fa_M}{(L - B)^2}}$$

and

$$\boxed{W_2 = \frac{2Fa_R}{B^2}.}$$

Type 3 right profile

For the right-oriented cantilever, the initial reverse-curvature length is

$$A = X_{12}.$$

The vertical components are

$$a_L = (Y_r - Y_l)\frac{A}{L}$$

and

$$a_M = (Y_r - Y_l)\frac{L - A}{L}.$$

The equivalent loads are

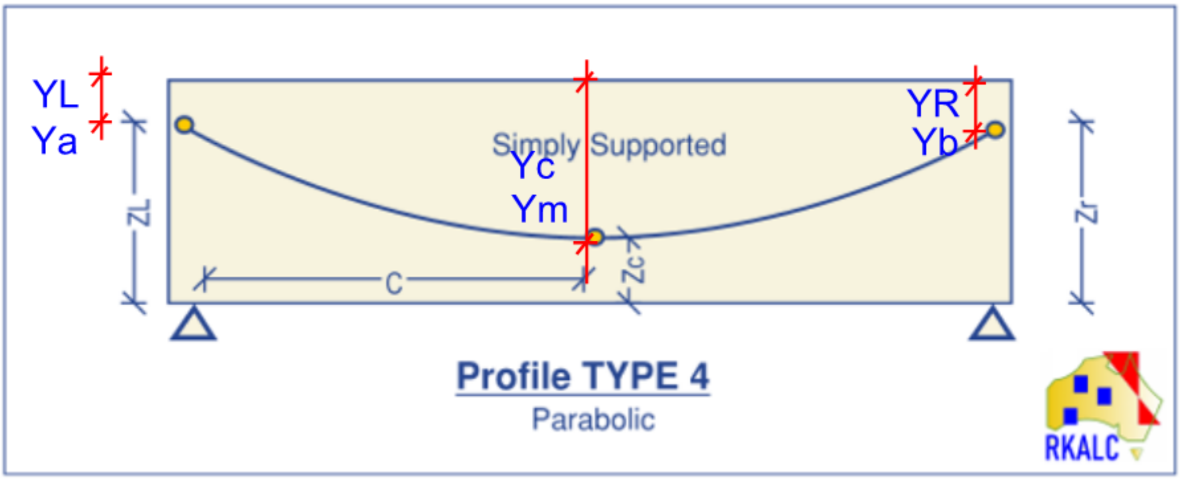
$$\boxed{W_1 = \frac{2Fa_L}{A^2}}$$

and

$$\boxed{W_2 = \frac{2Fa_M}{(L - A)^2}.}$$

The signs applied when entering these loads into the finite element table depend upon the cantilever orientation.

4.8. Type 4 Profile – Simply Supported Parabolic Tendon



Geometry

A Type 4 profile represents a simply supported parabolic tendon with no reverse-curvature region at either end.

The left parabola extends from

$$(0, Y_a)$$

to

$$(C, Y_c)$$

and is written as

$$y_2(x) = \frac{Y_a - Y_c}{C^2}(x - C)^2 + Y_c.$$

The right parabola extends from

$$(C, Y_c)$$

to

$$(L, Y_b)$$

and is written as

$$y_3(x) = \frac{Y_b - Y_c}{(L - C)^2}(x - C)^2 + Y_c.$$

Both segments are tangent at the internal profile point because

$$\left.\frac{dy_2}{dx}\right|_{x=C} = 0$$

and

$$\left.\frac{dy_3}{dx}\right|_{x=C} = 0.$$

Type 4 equivalent loads

The left-side drape is

$$a_2 = Y_m - Y_l$$

and the right-side drape is

$$a_3 = Y_m - Y_r.$$

The equivalent loads are

$$W_1 = 0,$$

$$\boxed{W_2 = \frac{2Fa_2}{C^2}}$$

$$\boxed{W_3 = \frac{2Fa_3}{D^2}}$$

and

$$W_4 = 0$$

where

$$D = L - C.$$

This formulation produces two uniform distributed loads, one over each side of the profile low or high point.

For the special symmetric case

$$C = \frac{L}{2}$$

and equal end elevations,

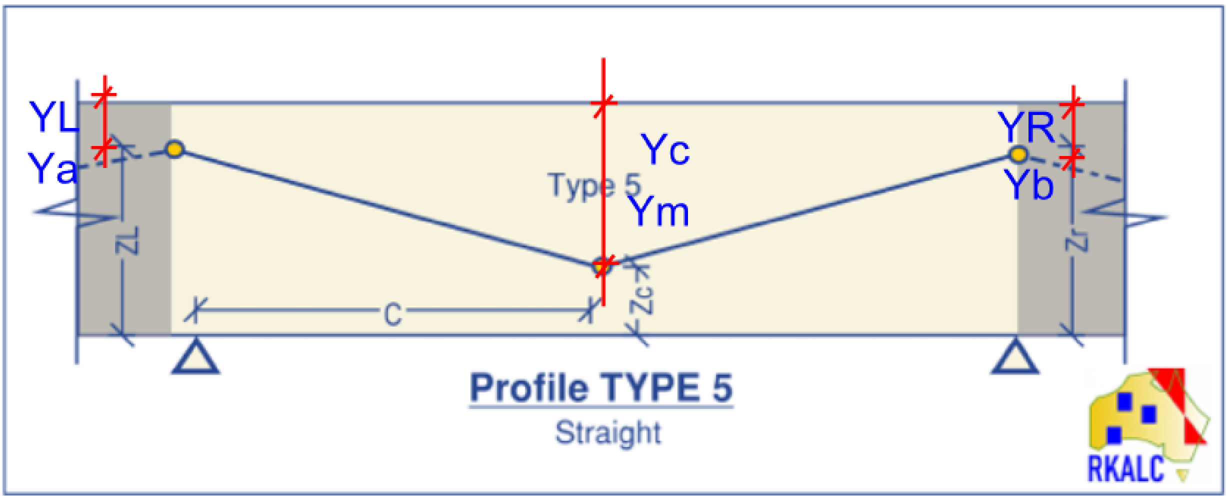
$$a_2 = a_3 = e,$$

each half-span load becomes

$$W = \frac{2Fe}{(L/2)^2} = \frac{8Fe}{L^2}.$$

This is the familiar load-balancing equation for a symmetric parabolic tendon.

4.9. Type 5 Profile – End-Span Harped Tendon



Geometry

The Type 5 profile consists of two straight tendon segments:

$$(0, Y_a) \rightarrow (C, Y_c) \rightarrow (L, Y_b).$$

The first-segment slope is

$$m_1 = \frac{Y_c - Y_a}{C}$$

and the second-segment slope is

$$m_2 = \frac{Y_b - Y_c}{L - C}.$$

Because each segment is straight,

$$\frac{d^2 y}{dx^2} = 0$$

within the segment lengths.

Therefore, no distributed equivalent load is generated between the profile points.

The prestress action occurs at the change in tendon direction at

$$x = C.$$

Equivalent concentrated load

PTKalc calculates the deviation of the internal point from the straight chord joining the two ends.

The chord elevation at $x = C$ is

$$Y_{\text{chord}} = Y_l + \frac{C}{L}(Y_r - Y_l).$$

The profile deviation is

$$a = Y_m - \left[Y_l + \frac{C}{L}(Y_r - Y_l)\right].$$

The equivalent concentrated force is then calculated as

$$\boxed{P_v = \frac{F\,a\,L}{C(L - C)}.}$$

Using

$$D = L - C,$$

this is the same as the implementation expression

$$P_v = \frac{FaL}{CD}.$$

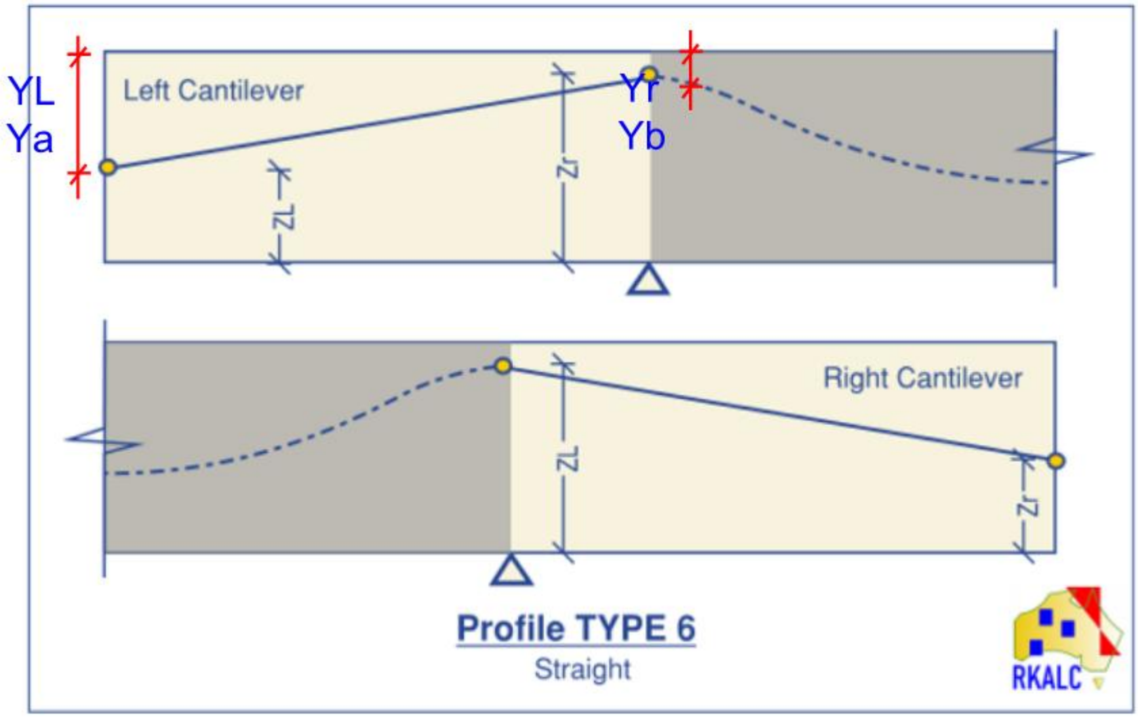
The concentrated load is entered at the central profile point. The adjacent finite element regions carry no distributed prestress loading.

This expression may also be interpreted from the small-angle change in tendon slope:

$$P_v \approx F(m_2 - m_1).$$

Substitution of the two line slopes leads to the same geometric relationship.

4.10. Type 6 Profile – Straight Inclined Tendon



Geometry

A Type 6 profile consists of a single inclined straight tendon segment extending from

$$(0, Y_a)$$

to

$$(L, Y_b).$$

The tendon slope is constant:

$$m = \frac{Y_b - Y_a}{L}.$$

Since the tendon curvature is zero,

$$\frac{d^2 y}{dx^2} = 0.$$

There is therefore no equivalent distributed loading along the member.

The inclined tendon does, however, introduce a vertical force component associated with its end direction.

Type 6 equivalent force

For the right-oriented profile, PTKalc calculates

$$\boxed{P_v = F\frac{Y_r - Y_l}{L}.}$$

For the left-oriented profile, the sign is reversed:

$$\boxed{P_v = F\frac{Y_l - Y_r}{L}.}$$

These expressions correspond to the small-angle vertical component

$$P_v \approx F\theta$$

with

$$\theta \approx \frac{\Delta y}{L}.$$

The equivalent force is entered into the finite element representation with orientation-dependent sign.

The supplied code uses separate Type 6 left and Type 6 right procedures to preserve the correct direction of the concentrated action.

4.11. Tendon Slope Change and Friction Loss

PTKalc evaluates tendon friction loss from the accumulated change in tendon slope.

For each parabolic segment, tendon slopes are sampled at a number of locations. The slope arrays are then processed by summing the absolute change between consecutive values:

$$\Theta = \sum_{i=2}^{n} |m_i - m_{i-1}|.$$

The total angular change for a complete profile is obtained by adding the contributions from all active profile segments:

$$\Theta_{\text{total}} = \Theta_1 + \Theta_2 + \Theta_3 + \Theta_4.$$

The effective force reduction is then calculated as

$$\boxed{\eta_f = \exp[-\mu(\Theta_{\text{total}} + kL)]}$$

where:

- μ is the tendon curvature-friction coefficient;
- k is the wobble coefficient;
- L is the tendon or span length in the length unit required by the loss equation.

In the implementation, the wobble term is written as

$$\frac{L}{1000}\beta_p,$$

so the force reduction becomes

$$\eta_f = \exp\left[-\mu\left(\Theta_{\text{total}} + \frac{L}{1000}\beta_p\right)\right].$$

The effective force used in the equivalent-load calculations is

$$\boxed{F_{\text{eff}} = F_0\eta_f.}$$

Thus, the distributed or concentrated equivalent loads are not generated from the unreduced input force. They are calculated using the force remaining after the profile-dependent friction adjustment.

4.12. Finite Element Load Segmentation

After calculating the equivalent loads, PTKalc passes them to the finite element model using the profile-segment boundaries.

The finite element load records include:

- element start coordinate;
- element end coordinate;
- uniform equivalent load;
- concentrated equivalent force where applicable;

- section inertia;
- section area;
- elastic modulus;
- beam width;
- beam depth.

The parabolic profiles are divided according to their curvature regions. In several cases, longer uniform-load zones are further divided into two finite element entries.

For example, a load zone extending from

$$x_1$$

to

$$x_2$$

may be divided at

$$x_m = \frac{x_1 + x_2}{2}.$$

The same equivalent uniform load is then applied over:

$$x_1 \leq x \leq x_m$$

and

$$x_m \leq x \leq x_2.$$

This subdivision does not alter the theoretical load intensity. Its purpose is to increase the number of analysis stations and improve the spatial representation of the tendon-induced loading in the finite element model.

The Type 5 profile uses zero distributed loading in the straight tendon regions and applies the calculated concentrated force at the internal deviation point.

The Type 6 profile applies the calculated vertical component as a concentrated element action rather than as a distributed load.

4.13. Summary of Profile Load Equations

The principal PTKalc equivalent-load equations may be summarised as follows.

For each parabolic segment:

$$\boxed{W_i = \frac{2F_{\text{eff}} a_i}{l_i^2}}$$

For a two-line harped profile:

$$\boxed{P_v = \frac{F_{\text{eff}} aL}{C(L - C)}}$$

For a single inclined straight profile:

$$\boxed{P_v = F_{\text{eff}} \frac{\Delta Y}{L}}$$

For friction reduction:

$$\boxed{F_{\text{eff}} = F_0 \exp\left[-\mu\left(\Theta_{\text{total}} + \frac{L}{1000}\beta_p\right)\right].}$$

The profile classification determines how the total tendon drape is separated into individual segment drapes a_i, how the corresponding lengths l_i are determined, and where each resulting load is applied to the structural model.

4.14. Computational Sequence

The complete PTKalc profile and equivalent-load procedure may be expressed as follows:

This sequence demonstrates that profile generation, friction loss and equivalent loading are integrated parts of one calculation. The profile geometry is not merely drawn after the analysis. It directly controls the force retained in the tendon and the loads introduced into the finite element model.

Read span geometry and tendon profile parameters

- Determine profile type
- Select left, right, internal, cantilever or simply supported formulation
- Transform tendon coordinates for graphical representation
- Construct each parabolic or straight profile segment
- Determine parabola transition coordinates
- Record tendon profile control points
- Sample tendon slopes along each curved segment
- Accumulate absolute slope changes
- Calculate friction reduction factor
- Reduce tendon force to the effective local force
- Resolve total tendon drape into individual segment drapes
- Calculate equivalent uniform or concentrated loads
- Divide the profile into finite element load regions
- Write the prestress loads and section properties to the FEA table
- Assemble the loads into the structural analysis model

5. Prestress in Load Cases and Load Combinations

PTKalc does not automatically apply the axial prestress force to every displayed stress result.

The selected result case is checked against:

- raw load cases;
- saved load combinations;
- the prestress load case identified as `PT`.

For the standalone prestress case:

$$\boldsymbol{P}_{\text{used}} = \boldsymbol{P}_{\text{av}}.$$

For a load combination containing a prestress factor $\boldsymbol{\gamma_{PT}}$:

$$\boxed{\boldsymbol{P}_{\text{used}} = \boldsymbol{\gamma_{PT}} \boldsymbol{P}_{\text{av}}.}$$

For a raw load case that does not contain prestress:

$$\boldsymbol{P}_{\text{used}} = \mathbf{0}.$$

This ensures that the direct prestress compression is included only when the selected result case contains the PT load action.

The section stresses are then determined from:

$$\boldsymbol{\sigma}_{\text{top}} = -\frac{\boldsymbol{P}_{\text{used}}}{\boldsymbol{A}} + \frac{\boldsymbol{M}\boldsymbol{y}_{\text{top}}}{\boldsymbol{I}}$$

and

$$\boldsymbol{\sigma}_{\text{bottom}} = -\frac{\boldsymbol{P}_{\text{used}}}{\boldsymbol{A}} - \frac{\boldsymbol{M}\boldsymbol{y}_{\text{bottom}}}{\boldsymbol{I}}.$$

For flanged sections, PTKalc uses the calculated section area, inertia and separate centroidal distances to the top and bottom fibres.

6. Profile Geometry Control and Automatic Load Balancing

Automatic load balancing assists in establishing a tendon geometry capable of producing a nominated equivalent load. The resulting profile remains subject to geometric, stress, strength, serviceability and detailing requirements and must be reviewed by the designer.

The attached code also contains an automatic load-balancing routine that should be documented separately from the loss calculations.

6.1. Target balanced load

The member self-weight is calculated as:

$$w_g = 25A$$

where:

- A is the section area in square metres;
- 25 is the adopted concrete unit weight in kN/m^3.

For a user-defined target percentage r, the target balancing load is:

$$\boxed{w_{\text{target}} = w_g \frac{r}{100}.}$$

6.2. Selection of tendon low-point depth

The programme the internal tendon level Z_c between:

$$30 \text{ mm}$$

and

$$0.8D$$

in 1 mm increments.

For each trial value, the Type 2 equivalent-load calculation is performed. The current code evaluates the absolute ratio of W2 to member self-weight and retains the lowest ratio encountered unless the early acceptance band is reached. This is not mathematically identical to minimising the error from the user-selected target percentage.

The intended optimisation target is the difference between the calculated balancing ratio and the user-selected target. For a true closest-match search, the comparison should minimise

$$\frac{|W_2|}{w_g} \approx \frac{r}{100}.$$

6.3. Adjustment of the low-point location

The programme subsequently varies the longitudinal coordinate C from:

$$0.2L$$

to

$$0.8L$$

in 10 mm increments.

For each trial position it evaluates:

$$\left|\frac{W_2}{W_3}\right|.$$

The early acceptance condition seeks to satisfy:

$$\mathbf{0.95} \le \left|\frac{W_2}{W_3}\right| \le \mathbf{1.00}.$$

When the acceptance band is reached, this moves the profile low point until the two principal parabolic load regions are approximately equal. If no trial reaches the band, the current fallback comparison retains the smallest |W2/W3| rather than the value closest to unity; a closest-match implementation should minimise ||W2/W3| - 1|.

The full automatic balancing sequence applies both routines twice:

- Adjust tendon depth
- Adjust longitudinal low-point position
- Adjust tendon depth again
- Adjust longitudinal low-point position again

The repeated sequence allows the vertical drape and horizontal low-point location to converge together.

7. Profile Geometry Limits and Continuity Corrections

The code enforces several limits before generating the tendon profile.

For the left profile elevation:

$$\mathbf{0.5D \le Z_l \le D - 30.}$$

For the internal low point:

$$\mathbf{30 \le Z_c \le 0.8D.}$$

For the right profile elevation, several conditional limits are used, including:

$$\mathbf{Z_r \le D - 30}$$

and minimum values related to the current and adjacent span depths.

The internal profile coordinate is restricted to:

$$\boxed{\mathbf{0.2L \le C \le 0.8L.}}$$

At changes in beam depth, the left elevation of a span is adjusted from the preceding span:

$$\mathbf{Z_{l,i} = Z_{r,i-1} - (D_{i-1} - D_i).}$$

This maintains the tendon's physical vertical position across a change in member depth rather than simply copying the same soffit offset.

That behaviour belongs in the tendon-continuity section of the final document

8. Prestressing Force, Friction Losses and Anchorage Draw-In

8.1. Introduction

After the tendon profile geometry has been generated, PTKalc determines the effective prestressing force available along the tendon before converting the profile into equivalent finite element loads. Unlike simplified procedures that assume a constant prestressing force throughout the structure, PTKalc models the progressive reduction in tendon force caused by friction and anchorage seating losses.

The procedure adopted by PTKalc follows the sequence shown below:

1. Determine the initial jacking force from the selected tendon properties.
2. Apply friction losses along each tendon profile.
3. Carry the reduced force forward into subsequent spans.
4. Calculate anchorage draw-in losses.
5. Reduce the equivalent balancing loads over the draw-in influence length.
6. Calculate the average prestressing force for section stress calculations.

This chapter describes each stage of that process.

8.2. Initial Jacking Force

The initial prestressing force is calculated from the tendon configuration selected by the user. The calculation depends on:

- strand size;
- strand area;
- stressing level;
- number of strands;
- number of tendons.

The code currently uses two strand sizes with nominal steel areas of:

- 98.6 mm^2
- 143 mm^2

The initial jacking force is therefore

$$P_0 = A_{ps} f_{pb} f_i N_s N_t$$

where

Symbol	Description
A_{ps}	Nominal strand area
f_{pb}	Prestressing steel stress
f_i	Stressing factor

Symbol	Description
N_s	Number of strands per tendon
N_t	Number of tendons

For the two strand sizes this becomes

$$P_0 = 98.6\, f_{pb} f_i N_s N_t$$

or

$$P_0 = 143\, f_{pb} f_i N_s N_t$$

This force represents the tendon force immediately after stressing and before any friction or anchorage losses have been applied.

The calculated value is stored internally and supplied to the tendon-profile routines for subsequent loss calculations.

8.3. Friction Losses

General

As the tendon passes through curved deviators and ducts, friction reduces the available prestressing force.

For profile Types 1 to 5, the supplied implementation evaluates a small-slope approximation to cumulative tendon angular change and calculates the remaining tendon force using the exponential friction equation

$$P = P_0 e^{-(\mu\theta + \beta_p x)}$$

where

Symbol	Description
P_0	Initial tendon force
P	Remaining tendon force
μ	Curvature friction coefficient
θ	Accumulated angular change
β_p	Wobble coefficient
x	Tendon length

PTKalc numerically samples the tendon profile and accumulates the absolute change in slope between successive points. For the shallow tendon slopes normally encountered in post-tensioned members, this is treated as a small-slope approximation to the accumulated angular change. The current routine does not explicitly evaluate the difference between successive arctangent angles.

For every span, the reduced tendon force leaving the profile routine becomes the starting force for the following span.

Consequently, friction losses accumulate naturally through continuous tendons extending across multiple spans.

The force entering span i is therefore

$$P_i = P_{i-1}\eta_i$$

where

$$\eta_i = e^{-(\mu\theta_i+\beta_p L_i)}$$

and the force after several spans becomes

$$P_n = P_0 \prod_{i=1}^{n} \eta_i$$

For profile Types 1 to 5, this approach carries the reduced force from one span into the next, modelling progressive friction loss from the stressing end. The supplied Type 6 routine uses the incoming force directly and does not apply a further profile-level friction reduction.

Implementation note: when the entered curvature-friction coefficient is zero, most profile routines substitute 0.001, whereas the supplied Type 4 routine substitutes 0.0001. This difference is retained here as a current implementation detail and should be reconciled in the code before publication as a uniform design rule.

8.4. Force Distribution Along the Structure

After friction has been evaluated for each span, PTKalc stores the remaining force for that span.

The program stores a span-by-span force history along the tendon rather than using one constant force for all spans.

These values are subsequently used for

- equivalent balancing loads;
- stress block calculations;
- axial compression;
- anchorage loss calculations.

The stored force diagram therefore represents the effective tendon force after friction losses but before anchorage seating losses.

8.5. Anchorage Draw-In Loss

General

Following completion of the friction calculations, PTKalc evaluates the reduction in tendon force caused by anchorage seating (draw-in).

When wedges seat during transfer of the stressing force, the tendon shortens slightly, reducing the prestressing force close to the anchorage.

Rather than reducing the force uniformly along the tendon, PTKalc assumes that this loss extends only over a finite influence length.

Beyond this distance the tendon force remains unchanged.

Average Force Gradient

The first step is to determine the average force gradient between the stressing end and the remote end of the tendon.

The program calculates

$$F_m = -\frac{P_{end} - P_{start}}{L}$$

where

Symbol	Description
F_m	Average force gradient
P_{start}	Prestressing force near the anchorage
P_{end}	Prestressing force at the end of the considered tendon
L	Effective tendon length

The negative sign converts the naturally decreasing tendon force into a positive loss gradient.

Draw-In Influence Length

The draw-in influence length is then calculated as

$$x_d = \sqrt{\frac{\delta_a E_p P}{f_{pb} f_i F_m}}$$

where

Symbol	Description
δ_a	Anchorage draw-in
E_p	Tendon elastic modulus
P	Prestressing force
F_m	Average force gradient

The adopted tendon elastic modulus is

$$E_p=195\,000\ {\rm MPa}$$

which is embedded directly within the implementation.

The resulting value represents the distance over which seating losses influence the tendon.

Maximum Force Reduction

The maximum reduction at the stressing end is

$$\Delta P = 2F_m x_d$$

This represents the reduction immediately adjacent to the anchorage.

The corresponding load reduction factor becomes

$$R = 1 - \frac{\Delta P}{P}$$

This factor is later applied to the equivalent balancing loads generated within the draw-in region.

Application to Finite Element Prestress Loads

Within the calculated draw-in influence length, PTKalc multiplies the LC1 uniform distributed prestress load of each traversed finite element by a single reduction factor. Member self-weight in LC2 is unchanged. In the supplied implementation, this post-processing operation modifies the distributed-load field only; concentrated Type 5 and Type 6 prestress actions are not altered by the same routine.

Force Variation Used for Stress Display

For the displayed tendon-force line and average span-force calculation, the draw-in correction is treated as varying linearly from its maximum value at the stressing end to zero at the draw-in influence limit. This differs from the constant reduction factor applied to finite element distributed loads within the same zone.

The local draw-in correction is represented by $\Delta P(x) = \Delta P_0 - 2F_m x$ for $0 \leq x \leq x\_d$. At $x = 0$ the loss is maximum, and at $x = x\_d$ the correction reduces to zero.

Implementation Checks

The draw-in calculation requires a positive and finite force gradient. Practical safeguards are required for a zero or negative gradient, an empty or single-entry force history, a negative square-root argument, and an influence length exceeding the available tendon length. These safeguards are not explicit in the supplied graphics routine and should be confirmed in the production calculation path.

9. Finite Element Representation of Prestress Actions

9.1. Introduction

Once the tendon geometry and effective prestressing force have been established, PTKalc converts the resulting tendon actions into load records suitable for structural analysis.

The finite element model does not contain a separate tendon element. Instead, the structural effects of the tendon are represented by equivalent actions applied directly to the beam elements. These actions include:

- uniform distributed loads generated by curved tendon segments;
- concentrated transverse forces generated by harped or inclined tendon segments;
- beam self-weight;
- section stiffness and geometry;
- support and column connectivity.

This arrangement allows the same analysis engine to solve both prestressed and conventionally loaded structures.

9.2. Finite Element Member Definition

Each finite element is defined by its two end coordinates:

$$(x_1, y_1)$$

and

$$(x_2, y_2).$$

For horizontal beam elements, the vertical coordinates are normally identical:

$$y_1 = y_2.$$

Vertical elements are also created where columns are included in the frame model.

Each member record includes the following structural properties:

$$\boldsymbol{E}$$

the elastic modulus,

$$\boldsymbol{I}$$

the second moment of area,

$$\boldsymbol{A}$$

the cross-sectional area,

$$\boldsymbol{b}$$

the analysis width, and

$$\boldsymbol{D}$$

the overall section depth.

The element record also includes distributed and concentrated load data for the available load cases.

9.3. Section Properties Assigned to Elements

PTKalc calculates section properties before creating the finite element records.

For a rectangular section of width $\boldsymbol{b}$ and depth $\boldsymbol{D}$,

$$\boldsymbol{A = bD}$$

and

$$\boldsymbol{I = \frac{bD^3}{12}}.$$

The centroid is located at

$$\boldsymbol{y_t = y_b = \frac{D}{2}}.$$

For T- and L-shaped sections, the section is divided into:

- a flange rectangle;
- a web rectangle below the flange.

The flange area is

$$\boldsymbol{A_f = b_f t_f}$$

and the web area is

$$\boldsymbol{A_w = b_w(D - t_f)}.$$

The total area is

$$\boldsymbol{A = A_f + A_w}.$$

The centroid measured from the top of the section is

$$\boldsymbol{\bar{y} = \frac{A_f y_f + A_w y_w}{A}}.$$

The total second moment of area is calculated using the parallel-axis theorem:

$$\boldsymbol{I = I_f + A_f(\bar{y} - y_f)^2 + I_w + A_w(y_w - \bar{y})^2}.$$

The calculated area and inertia are passed into every finite element associated with that span.

This means the equivalent prestress loads and the structural stiffness both reflect the selected section geometry.

9.4. Prestress Load Case

PTKalc stores the equivalent tendon loading in the first load-case position of each element record.

The code writes the prestress uniform load into:

$$\boldsymbol{LC1}$$

while the member self-weight is stored separately in:

$$\boldsymbol{LC2}.$$

Thus:

$$\boldsymbol{LC1} = \text{equivalent prestress load}$$

and

$$LC2 = \text{member self-weight.}$$

For horizontal members, the self-weight is calculated as:

$$\boxed{w_g = -25A}$$

where:

- A is the section area in square metres;
- 25 is the adopted concrete unit weight in kN/m^3.

The negative sign follows the downward load convention used in the model.

The prestress load in $LC1$ may be either positive or negative depending upon the tendon curvature and the element orientation.

9.5. Distributed Prestress Loads

For parabolic tendon profiles, each constant-curvature region generates a uniform equivalent load.

The general expression is:

$$w_i = \frac{2P_{\text{eff}}a_i}{l_i^2}.$$

PTKalc assigns the calculated value to the finite elements covering that profile region.

For example, a Type 2 profile may produce four load regions:

$$W_1, \quad -W_2, \quad -W_3, \quad W_4.$$

The corresponding finite element regions are:

$$0 \le x \le A,$$
$$A \le x \le C,$$
$$C \le x \le C + D - B,$$

and

$$C + D - B \le x \le L.$$

The profile calculation therefore determines the magnitude of each equivalent load, while the element-creation call controls its final algebraic direction.

9.6. Element Subdivision

PTKalc frequently divides one theoretical equivalent-load region into two finite element records.

For a load region extending from x_1 to x_2, the midpoint is

$$x_m = \frac{x_1 + x_2}{2}.$$

The same uniform load is then applied over:

$$x_1 \le x \le x_m$$

and

$$x_m \leq x \leq x_2.$$

For example, the left parabola of a Type 1 profile may be represented by:

$$0 \leq x \leq \frac{C}{2}$$

and

$$\frac{C}{2} \leq x \leq C.$$

The load intensity is unchanged:

$$w_{1a} = w_{1b} = W_2.$$

The purpose of this subdivision is not to modify the theoretical equivalent loading. It introduces additional finite element nodes and therefore provides more analysis stations within the span.

This improves:

- moment resolution;
- shear resolution;
- deflection interpolation;
- identification of local extrema;
- subsequent cracked-stiffness assignment.

9.7. Concentrated Prestress Loads

Straight tendon segments have zero curvature and therefore produce no distributed loading within their lengths.

Where a straight tendon changes direction, PTKalc represents the prestress effect by a concentrated transverse load.

For a Type 5 profile, the concentrated force is:

$$P_v = \frac{P_{\mathrm{eff}} a L}{C(L-C)}.$$

The finite element representation uses zero distributed load on the adjacent segments:

$$w = 0$$

and applies the concentrated force at the profile deviation point.

The Type 5 span is divided into four finite element regions:

$$0 \rightarrow \frac{C}{2},$$
$$\frac{C}{2} \rightarrow C,$$
$$C \rightarrow C + \frac{L-C}{2},$$

and

$$C + \frac{L-C}{2} \rightarrow L.$$

The point force is applied through the element load fields associated with the deviation location.

For a Type 6 inclined straight tendon, the vertical force component is calculated as:

$$P_v = P_{\text{eff}} \frac{\Delta y}{L}.$$

This force is entered as a concentrated element action with the sign determined by the left- or right-oriented profile procedure.

9.8. Point-Load Position

The `PTKalc` routine receives both:

$$P$$

the point-load magnitude, and

$$X_p$$

the point-load position.

These values are written into the point-load columns of the finite element table.

The point-load position is measured relative to the element definition used by the solver.

The profile routines subdivide spans at geometric transition points and pass both point-load magnitude and a position parameter to the finite element table. The precise interpretation of that position parameter is governed by the structural solver and should be verified against its load convention.

The subdivision provides additional analysis stations around profile changes, but it does not by itself establish the solver's point-load coordinate convention.

9.9. Anchorage Draw-In Modification of Element Loads

Following initial element creation, PTKalc applies the anchorage draw-in reduction to the prestress load case.

The reduction is applied only to elements encountered before the accumulated member length exceeds the draw-in influence length.

The self-weight load case is not modified.

Therefore:

$$LC1_{\text{new}} = R_d LC1_{\text{old}}$$

while

$$LC2_{\text{new}} = LC2_{\text{old}}.$$

This preserves the distinction between tendon-induced loading and gravity loading.

9.10. Support Representation

The structural model includes support records, each support record contains restraint flags for:

- horizontal translation;
- vertical translation;

- rotation.

The restraint records used by the current implementation include the following flag combinations:

Pin support

$$u_x = 0, \qquad u_y = 0, \qquad \theta \text{ free.}$$

This is stored using:

$$H = 1, \qquad V = 1, \qquad M = 0.$$

Support with horizontal restraint active

$$u_x = 0, \qquad u_y \text{ free}, \qquad \theta \text{ free}$$

or the orientation-specific equivalent used by the frame model.

This is stored as:

$$H = 1, \qquad V = 0, \qquad M = 0.$$

Where columns are present, vertical finite elements are created between the beam and the remote support node. The column stiffness is therefore explicitly included rather than replacing the column by a simple support condition.

9.11. Column Elements

Column members are created using the same finite element table as the beam members.

For a rectangular column, the area is:

$$A_c = b_c h_c.$$

The column inertia used in the frame plane is:

$$I_c = \frac{h_c b_c^3}{12}$$

for the orientation represented in the code.

The column elements receive zero distributed prestress loading:

$$w = 0$$

and zero concentrated tendon load:

$$P = 0.$$

Their function is to contribute stiffness and transfer beam actions to the support nodes.

This is important for continuous post-tensioned frames because the secondary prestress actions depend upon frame restraint. The equivalent tendon loads act on the beam, but the resulting reactions and moments are influenced by the stiffness of the connected columns.

9.12. Primary and Secondary Prestress Effects

The equivalent tendon loads directly represent the primary structural action of the tendon profile.

For a statically determinate member, these loads reproduce the familiar primary prestress moment:

$$M_p(x) = Pe(x).$$

In a continuous or otherwise restrained frame, the equivalent loads also generate support reactions and restraint moments.

The total response obtained from the finite element solution includes:

$$M_{\text{total}} = M_{\text{primary}} + M_{\text{secondary}}.$$

The secondary moment is therefore:

$$\boxed{M_{\text{secondary}} = M_{\text{analysis}} - Pe}$$

subject to the adopted sign convention.

PTKalc does not need to apply a separate manually calculated secondary-moment load. Secondary effects arise automatically from the structural analysis because the equivalent tendon loads are applied to the complete restrained frame.

This is the standard engineering interpretation of the response obtained by applying the equivalent tendon loads to the restrained frame. The explicit subtraction of primary from total moment is not performed in the supplied graphics routine.

9.13. Load Visualisation

The finite element prestress loads stored in the first load case are also used to draw the balancing-load diagram.

For every beam element, PTKalc reads the equivalent distributed load and draws a set of arrows perpendicular to the member axis.

The arrow direction is based upon the load sign.

Positive and negative load regions are visually distinguished, and the load intensity is labelled numerically.

For upward balancing loads, the program additionally displays the ratio of the balancing load to member self-weight:

$$\boxed{r_b = \frac{w_{PT}}{w_g} \times 100\%.}$$

This is shown as a percentage of own weight.

The graphical output therefore acts as a direct verification of the equivalent-load model stored in the finite element table.

9.14. Element Metadata and Variable Sections

Each horizontal finite element receives the metadata of its parent beam-schedule span.

The member identifier:

$$F_i$$

is used to associate the finite element with schedule row i.

This is particularly important where:

- adjacent spans have different depths;
- T-sections or L-sections are used;
- the beam area and inertia vary by span;

- stress calculations require the correct local centroid and section modulus.

PTKalc assigns the selected section metadata to every finite element belonging to that span, rather than determining the section from geometric midpoint alone.

This ensures that the last element of a span and elements located at common support coordinates retain the correct section assignment.

9.15. Computational Sequence

The finite element load-generation process can be summarised as follows:

- Read selected beam and column sections
- Calculate area, inertia, centroid and section depth
- Generate tendon profile for the span
- Calculate friction-reduced tendon force
- Calculate each equivalent distributed or concentrated load
- Determine profile transition coordinates
- Divide the span at load-region boundaries
- Subdivide longer profile regions where additional resolution is required
- Create beam finite elements
- Store prestress loading in LC1
- Store self-weight in LC2
- Create column finite elements where present
- Create support restraint records
- Assign section metadata to every horizontal element
- Calculate anchorage draw-in influence length
- Reduce LC1 loads inside the draw-in region
- Pass the complete table to the frame solver

Summary

PTKalc represents prestressing within the structural model by applying equivalent tendon actions directly to conventional beam finite elements.

The procedure:

- assigns the actual area and inertia of each selected section;
- divides each tendon profile into constant-load regions;
- applies uniform loads to parabolic regions;
- applies concentrated forces at tendon deviations;
- subdivides long regions to increase analysis resolution;
- stores prestress and self-weight in separate load cases;
- includes column stiffness and support restraints;
- modifies prestress loads for anchorage draw-in;
- allows secondary prestress effects to arise automatically from frame restraint.

The resulting finite element model contains all information required to determine reactions, shear forces, bending moments and deflections caused by the tendon profile and the surrounding structural system.

10. PTKalc Analysis and Design Methodology

10.1. Overview

PTKalc is a finite element analysis and design program developed for the analysis of reinforced and prestressed concrete beam structures. The program combines structural analysis with section-level design calculations to evaluate both the ultimate strength and serviceability performance of concrete members in accordance with the requirements of Australian Standards.

Unlike conventional beam design programs that perform structural analysis and member design as largely independent processes, PTKalc integrates these calculations into a unified analysis procedure. Internal forces obtained from the finite element model are used to determine the reinforcement requirements and serviceability characteristics of each beam section. These calculated section properties are subsequently incorporated into a second stage of structural analysis, allowing the effects of cracking, prestressing, creep and shrinkage to influence the predicted structural behaviour.

This approach provides a realistic representation of reinforced and prestressed concrete structures by recognising that member stiffness varies continuously along the length of a beam. Rather than assuming a constant stiffness for an entire member, PTKalc evaluates the behaviour of individual finite elements based on their local loading conditions and reinforcement characteristics, resulting in more representative predictions of structural deformation and force redistribution.

10.2. Analysis Philosophy

The analysis methodology adopted by PTKalc is based on the principle that the behaviour of reinforced concrete structures cannot be represented accurately using a single constant flexural stiffness. As loading increases, individual regions of a beam may remain uncracked, partially cracked or fully cracked, while prestressing, reinforcement, creep and shrinkage each contribute to the overall stiffness of the structure.

To account for these effects, PTKalc performs the structural analysis in a sequence of interconnected stages. An initial elastic analysis establishes the distribution of forces and moments throughout the structure. These actions are then used to evaluate the response of every beam section, including its flexural capacity, cracking behaviour and effective stiffness. The resulting stiffness values are assigned to the corresponding finite elements, and the structure is analysed again using this updated stiffness distribution to determine long-term structural behaviour.

By combining structural analysis with detailed section calculations, PTKalc captures the interaction between member stiffness and structural response. Regions experiencing significant cracking become less stiff and therefore attract reduced bending moments, while stiffer regions continue to carry a greater proportion of the applied loading. This redistribution occurs naturally through the finite element analysis without requiring empirical adjustments or simplified assumptions regarding member behaviour.

10.3. Scope of the Methodology

The methodology described in this document applies to reinforced and prestressed concrete beams analysed using the PTKalc finite element engine. The procedures encompass the determination of bending moments and internal actions, flexural reinforcement design, cracking assessment, effective stiffness evaluation, and the calculation of short-term and long-term deflections.

The analysis procedures have been developed to provide a practical engineering representation of structural behaviour while remaining computationally efficient for the analysis of continuous beams and multi-span structures. Particular emphasis has been placed on modelling the influence of local cracking, prestressing and time-dependent material effects on member stiffness, allowing deformation predictions to reflect the changing behaviour of reinforced concrete throughout its service life.

The following sections describe the principal stages of the PTKalc analysis procedure, beginning with the overall computational workflow, followed by the flexural design methodology and the serviceability and deflection calculations.

11. Overall Analysis Procedure

11.1. General

PTKalc employs a staged finite element analysis procedure to determine the strength and serviceability performance of reinforced and prestressed concrete beam structures. Rather than relying on a single structural analysis using constant member properties, the program progressively refines the structural model by incorporating the effects of reinforcement, prestressing, cracking and long-term material behaviour into the calculated member stiffnesses.

The overall analysis consists of two principal stages. The first establishes the distribution of internal forces throughout the structure using the gross section properties of each member. These forces are then used to evaluate the structural behaviour of every beam section individually. Following the section calculations, updated stiffness properties are assigned to each finite element, allowing a second structural analysis to predict the long-term response of the structure.

This staged approach allows the structural model to respond naturally to local variations in member behaviour while maintaining compatibility throughout the complete structure.

11.2. Finite Element Modelling

The structural model is assembled using beam finite elements representing the geometry, support conditions and connectivity of the structure. Material properties and section properties are assigned to each member together with any applicable prestressing tendons and externally applied loads.

Each beam is divided into multiple finite elements to ensure that changes in bending moment, cracking and stiffness can be represented along the member length. This discretisation enables local structural behaviour to be captured without requiring individual members to be subdivided manually by the user.

All loads, including permanent actions, imposed actions and equivalent prestressing loads, are incorporated into the finite element model. Standard matrix analysis techniques are then used to determine nodal displacements, support reactions and internal member actions.

11.3. Initial Structural Analysis

The first stage of analysis is performed using the gross elastic properties of each concrete section. At this stage the structure is assumed to remain uncracked, providing a consistent basis for determining the distribution of internal forces generated by the applied loading.

The analysis produces the bending moments, shear forces, axial forces and support reactions throughout the structure. These internal actions form the basis for the subsequent section calculations and are evaluated at each finite element location.

Because the initial analysis is performed before cracking is considered, it represents the elastic behaviour of the structure immediately after loading and provides the reference force distribution from which reinforcement requirements and serviceability behaviour are assessed.

11.4. Section Assessment

Following completion of the initial structural analysis, each beam section is evaluated independently using the calculated internal forces.

The section calculations determine the reinforcement required to satisfy the applied bending moments and assess the corresponding serviceability behaviour of the section. This includes evaluation of the cracking state, effective flexural stiffness and the influence of prestressing, creep and shrinkage on the local structural response.

Because the section calculations are performed at the two ends of each finite element and then averaged for element assignment, the stiffness can vary from one element to the next in accordance with the local loading conditions, rather than being assumed uniform over the complete member.

11.5. Stiffness Updating

Once the section calculations have been completed, the effective flexural stiffness of each finite element is determined.

Unlike conventional methods that assign a single stiffness to an entire member, PTKalc allows each finite element to possess an individual stiffness corresponding to its calculated cracking state and reinforcement characteristics. Consequently, adjacent elements within the same beam may exhibit different stiffness values depending upon the local bending moment and section response.

This element-by-element representation provides a more realistic simulation of reinforced concrete behaviour by recognising that cracking can differ between adjacent portions of the member rather than developing uniformly over its entire length.

11.6. Long-Term Structural Analysis

The updated stiffness distribution is incorporated into a second finite element analysis representing the long-term behaviour of the structure.

During this stage the effects of cracking, prestressing, creep and shrinkage are reflected in the stiffness of each element, allowing the structural response to develop naturally under the revised member properties. The resulting bending moments and deflections therefore account for the interaction between local section behaviour and global structural compatibility.

Because the finite element stiffness matrix is reconstructed using the calculated effective stiffnesses, the redistribution of internal forces occurs automatically as part of the structural analysis. Regions that have experienced significant cracking contribute less stiffness to the structure, while less highly stressed regions continue to attract a greater proportion of the applied loading.

11.7. Final Structural Response

The second-stage analysis produces the final predictions of member deflections, internal forces and support reactions under service conditions.

These results represent the combined influence of structural continuity, reinforcement, prestressing, cracking and long-term material behaviour. By incorporating the calculated section stiffness directly into the finite element analysis, PTKalc provides a realistic representation of structural behaviour while maintaining compatibility across the complete structure.

This methodology also provides a consistent framework for both reinforced and prestressed concrete members, allowing a common analysis procedure to be applied regardless of section geometry or reinforcement arrangement.

11.8. Overall Analysis Workflow

Structural Model →

Finite Element Mesh Generation →

Initial Elastic Structural Analysis →

Recovery of Internal Forces →

Section Assessment • Flexural Design • Cracking Assessment • Prestress Effects • Effective Stiffness →

Update Element Stiffnesses →

Long-Term Finite Element Analysis →

Final Internal Forces and Deflections

12. Flexural Design Methodology (AS3600-2018)

12.1. General

Following completion of the initial structural analysis, PTKalc performs a detailed flexural assessment of every beam section using the bending moments obtained from the finite element analysis. The objective of this stage is to determine the reinforcement required to satisfy the design actions while simultaneously establishing the section properties required for subsequent serviceability calculations.

Unlike conventional design procedures where reinforcement is determined only at a limited number of critical locations, PTKalc evaluates section behaviour at the finite-element locations used by the structural model. This provides a closely spaced representation of reinforcement demand and enables the local member behaviour to be carried into the subsequent serviceability analysis.

The flexural design procedure is applicable to reinforced concrete and prestressed concrete members and accommodates rectangular, T-shaped and L-shaped beam sections using a common calculation methodology.

12.2. Section Behaviour

Each section is analysed using the principles of strain compatibility and internal force equilibrium. Plane sections are assumed to remain plane after bending, resulting in a linear strain distribution through the depth of the section.

The compressive forces developed within the concrete are balanced by the tensile forces in the reinforcement and prestressing tendons together with any compression reinforcement present within the section. Equilibrium between these internal forces establishes the position of the neutral axis and the corresponding flexural resistance of the member.

This approach allows reinforced and prestressed concrete sections to be analysed using a consistent methodology irrespective of the reinforcement arrangement or section geometry.

12.3. Section Geometry

PTKalc supports the analysis of rectangular, T-shaped and L-shaped beam sections. Each section is represented using its actual geometric properties so that the contribution of flanges and webs is correctly incorporated into both the strength and serviceability calculations.

For flanged sections, the compression zone may develop entirely within the flange or extend into the web depending upon the applied bending moment and the calculated neutral axis depth. The program automatically identifies the governing compression region and determines the corresponding compression force without requiring separate calculation procedures for different section types.

This unified approach ensures that all supported beam geometries are treated consistently throughout the analysis.

12.4. Reinforcement Assessment

For each design section, PTKalc evaluates the reinforcement required to resist the applied design moment obtained from the structural analysis.

The program first determines whether the applied loading can be resisted using a singly reinforced section. Where the required moment exceeds the capacity achievable within the permissible neutral axis limits, additional compression reinforcement is incorporated into the design and the section is treated as doubly reinforced.

This procedure enables the program to produce practical reinforcement layouts while satisfying the ductility and strength requirements of the adopted design standard.

Where prestressing tendons are present, their contribution to the internal force equilibrium is included directly in the section analysis. The effective prestressing force and tendon eccentricity therefore influence both the required reinforcement and the resulting flexural capacity.

The tendon depth used in the equilibrium calculation is referenced from the active compression face and is therefore transformed according to the direction of bending.

12.5. Ultimate Capacity

Once a satisfactory reinforcement arrangement has been established, the ultimate flexural capacity of the section is evaluated.

The analysis verifies that equilibrium exists between the concrete compression block, reinforcing steel and prestressing tendons while satisfying the strength reduction requirements of the applicable design standard.

The calculated ultimate capacity is then compared with the design bending moment to confirm that adequate strength has been provided.

The design capacity includes a strength reduction factor determined by the detailed section solver from the neutral-axis ratio, rather than a single fixed factor applied to all sections.

Because the calculations are performed using the actual section geometry and reinforcement arrangement, the resulting flexural capacity reflects the behaviour of the completed section rather than relying upon simplified design charts or approximate relationships.

12.6. Reinforcement Distribution

Since flexural calculations are performed at each finite element station, reinforcement demand is evaluated at closely spaced locations along the length of every beam.

This enables PTKalc to identify regions of maximum reinforcement demand together with transitions between positive and negative bending. The resulting reinforcement distribution provides the basis for practical detailing while also supplying the reinforcement information required for the subsequent serviceability calculations.

The reinforcement determined during this stage therefore serves two purposes. It establishes compliance with ultimate strength requirements and provides the reinforcement characteristics needed to evaluate cracking behaviour and effective flexural stiffness.

12.7. Relationship to Serviceability

The flexural design stage forms the link between structural analysis and serviceability assessment.

Once the reinforcement for each section has been determined, PTKalc proceeds to evaluate the corresponding service behaviour under working loads. At each finite-element end, the program determines the cracking moment, neutral-axis position, cracked section properties and effective flexural stiffness using the reinforcement arrangement established during the flexural design process.

These calculated stiffnesses form the basis of the second-stage finite element analysis described in the following chapter.

Accordingly, the reinforcement design is not treated as an isolated calculation but as an integral component of the overall analysis methodology. The reinforcement determined during the strength assessment directly influences the stiffness assigned to each finite element and therefore contributes to the prediction of long-term structural behaviour.

12.8. Governing Design Relationships

$$d = D - \text{cover}$$
$$\alpha_2 = \max(0.67,\ 0.85 - 0.0015f'_c)$$
$$\gamma = \max(0.67,\ 0.97 - 0.0025f'_c)$$
$$a = \gamma x$$
$$x_{max} = 0.36d$$
$$A_{s,max} = 0.04b_w D$$

For reinforced concrete sections, the minimum tension reinforcement is:

$$A_{s,min} = 0.24\left(\frac{D}{d}\right)^2 \frac{0.6\sqrt{f'_c}}{f_{sy}} db_w$$

which may be written in simplified form as:

$$A_{s,min} = 0.144\left(\frac{D}{d}\right)^2 \frac{\sqrt{f'_c}}{f_{sy}} db_w$$

For prestressed sections, this particular reinforced-concrete minimum area is not imposed by the flexural design procedure.

The factored prestress force entering ultimate equilibrium is:

$$P_t = \max(0,\ \phi_{PT}P \times 1000)$$

A singly reinforced solution is first sought within the adopted neutral-axis limit:

$$x \le 0.36d$$

Compression reinforcement is introduced where a singly reinforced solution cannot satisfy the design action within this limit. Where minimum tension reinforcement governs, force equilibrium and flexural capacity are recalculated using the adopted reinforcement. The maximum permitted area of either tension or compression reinforcement is:

$$A_s \le 0.04b_w D$$

12.9. Flexural Design Workflow

Finite element bending moments →

Determine section geometry →

Establish strain compatibility →

Calculate internal force equilibrium →

Determine reinforcement requirements →

Verify ultimate capacity →

Establish reinforcement distribution →

Provide reinforcement data for serviceability calculations

13. Shear Design Methodology (AS3600-2018)

13.1. General

PTKalc performs shear design at discrete locations along the member using the design shear force obtained from the governing ultimate load combination. The shear design procedure evaluates the concrete contribution to shear resistance, the required transverse reinforcement, the longitudinal strain condition of the member and the maximum permissible shear capacity.

The procedure is based on a variable-angle shear model in which the concrete shear contribution and the inclination of the compression field are dependent on the calculated longitudinal strain at the section. Flexural reinforcement requirements determined by the flexural design procedure are therefore incorporated into the shear assessment.

Prestress actions contained within the finite element load combinations are retained in the design shear force. The vertical component of tendon force is consequently not separately deducted within the shear design calculation, avoiding duplication of the prestress contribution already represented in the structural analysis.

13.2. Effective Shear Depth

The effective shear depth used in the shear calculation is determined from the overall section depth and the effective flexural depth:

$$d_v = \max(0.72D,\ 0.9d)$$

where:

D = overall section depth;
d = effective depth to the longitudinal reinforcement; and
d_v = effective shear depth.

The effective flexural depth is initially determined from the overall depth and reinforcement cover.

13.3. Minimum Shear Reinforcement

PTKalc establishes the minimum transverse reinforcement requirement independently of the calculated shear demand. The minimum shear reinforcement ratio used by the shear design procedure is:

$$\frac{A_{sv}}{s} = \frac{0.08\sqrt{f_c'}\,b_w}{f_{sy}}$$

where:

A_{sv} = area of shear reinforcement;
s = longitudinal spacing of shear reinforcement;
f_c' = characteristic concrete compressive strength;
b_w = effective web width; and
f_{sy} = yield strength of shear reinforcement.

Where the calculated reinforcement requirement is less than this value, PTKalc reports the minimum reinforcement requirement.

13.4. Longitudinal Strain

The longitudinal strain of the member is used to determine the concrete shear parameter and the inclination of the compression field.

PTKalc first performs the flexural design calculation at the section and obtains the required longitudinal reinforcement. The longitudinal strain parameter is then calculated using the concurrent bending moment, shear force, axial force and longitudinal reinforcement.

The calculated strain is limited to a maximum value of:

$$\varepsilon_v \le 0.003$$

and negative calculated values are taken as zero.

This establishes a direct interaction between the flexural condition of the section and its calculated shear resistance. Sections experiencing greater longitudinal tensile strain consequently develop a reduced concrete contribution and a modified compression-field angle.

13.5. Concrete Shear Parameter

The concrete shear parameter k_v is determined from the calculated longitudinal strain:

$$k_v = \frac{0.4}{1 + 1500\varepsilon_v}$$

As the longitudinal tensile strain increases, k_v reduces, thereby reducing the calculated concrete contribution to shear resistance.

13.6. Compression-Field Angle

The inclination of the compression field is calculated from:

$$\theta_v = (29 + 7000\varepsilon_v)^\circ$$

The resulting angle is subsequently used in both the maximum shear capacity calculation and the determination of the required transverse reinforcement.

The use of a strain-dependent compression-field angle allows the shear design calculation to reflect the changing flexural and axial condition of the member rather than adopting a single fixed truss angle throughout the beam.

13.7. Concrete Contribution to Shear Capacity

The design concrete contribution to shear capacity is calculated as:

$$\phi V_{uc} = 0.75\, k_v\, b_w\, d_v \sqrt{f_c'}$$

with the appropriate unit conversion applied internally by PTKalc.

The corresponding concrete shear stress may be expressed as:

$$v_c = \frac{\phi V_{uc}}{b_w d_v}$$

The applied design shear is compared with this capacity to establish whether transverse reinforcement is required beyond the minimum reinforcement provision.

13.8. Required Shear Reinforcement

Where the equivalent design shear does not exceed the concrete shear capacity, PTKalc adopts the minimum shear reinforcement requirement.

Where:

$$V^* > \phi V_{uc}$$

the required transverse reinforcement is calculated from the shear demand remaining after the concrete contribution has been deducted:

$$\frac{A_{sv}}{s} = \frac{(V^* - \phi V_{uc})}{0.75\, f_{sy}\, d_v \cot\theta_v}$$

with the appropriate force and dimensional unit conversions applied internally.

The calculated reinforcement is compared with the minimum shear reinforcement requirement, and the greater requirement governs.

The reported shear reinforcement envelope therefore represents the transverse reinforcement required to satisfy the governing ultimate shear demand at each design location along the member.

13.9. Maximum Shear Capacity

In addition to calculating the required transverse reinforcement, PTKalc checks the section against the maximum permissible shear resistance.

The maximum shear force is evaluated as:

$$V_{u,\max} = 0.55 f_c' b_w d_v \left[\frac{\cot\theta_v}{1 + \cot^2\theta_v}\right]$$

and the corresponding design shear stress is obtained after application of the capacity reduction factor.

PTKalc independently calculates the applied shear stress at the section and compares this with the maximum permissible value.

Where the applied shear stress exceeds the maximum permissible shear stress, the section is reported as:

Shear Failure

This check represents a section-capacity limit and cannot be satisfied simply by increasing the quantity of transverse reinforcement.

13.10. Prestress Effects

Prestressing influences the shear design through the structural actions generated by the prestress load case and its participation in the governing load combination.

PTKalc does not separately subtract the vertical component of tendon force from the design shear within the Shear Designer procedure. The equivalent prestress loads generated from the tendon profile have already been incorporated into the finite element analysis and consequently into the shear-force diagram used for design.

Separately introducing the tendon vertical component within the shear design calculation would therefore duplicate an effect already contained within the applied shear action.

The current shear design implementation also does not directly credit the tendon force as longitudinal tensile reinforcement in the calculation of longitudinal shear strain.

13.11. Torsion

The shear design procedure contains provision for interaction between shear and torsion. However, torsional demand is presently set to zero within PTKalc.

Accordingly, the current implementation performs the design as a shear assessment without an applied torsional action. The torsional terms retained within the calculation procedure do not contribute to the reported shear reinforcement for the current application.

13.12. Shear Reinforcement Distribution

Following determination of the required A_{sv}/s, PTKalc evaluates the transverse distribution of stirrup legs across the section width.

A minimum of two legs is initially considered. Where the transverse distance between legs exceeds the permitted spacing criterion, additional legs are introduced across the width of the section.

The shear reinforcement requirement calculated at successive design locations is used to establish the shear reinforcement envelope presented in the PTKalc results.

13.13. Shear Design Sequence

The shear design procedure may therefore be summarised as:

Ultimate FE shear and moment actions
↓
Determine section geometry and effective shear depth
↓
Obtain longitudinal reinforcement from flexural design
↓
Calculate longitudinal strain ε_v
↓
Determine k_v and compression-field angle θ_v
↓
Calculate concrete shear capacity ϕV_{uc}
↓
Calculate required A_{sv}/s
↓

Apply minimum shear reinforcement requirement
↓
Check maximum shear capacity
↓
Establish shear reinforcement envelope

This methodology couples the shear assessment to the flexural state of the member while retaining the shear actions, including the global effects of prestressing, determined directly from the finite element analysis.

14. Serviceability and Deflection Analysis (AS3600-2018)

14.1. General

Serviceability behaviour is governed primarily by the stiffness of the structural members under working loads. Unlike ultimate strength calculations, where the objective is to determine the load-carrying capacity of a section, serviceability analysis seeks to predict the deformations that occur throughout the life of the structure.

The stiffness of reinforced concrete is not constant. As the applied moment increases, cracking develops within the tensile zone, reducing the effective flexural rigidity of the member. Additional reductions occur over time due to concrete creep. Shrinkage changes the internal stress state and may contribute to additional deformation; in the present PTKalc procedure, its influence is incorporated through the cracking and effective-stiffness assessment. Prestressing reduces tensile stress and delays the onset of cracking.

PTKalc models these mechanisms by evaluating the section stiffness at each finite-element end, assigning one effective stiffness to each element, and then performing a second structural analysis using the updated member properties.

14.2. Cracking Moment

The first step is to determine whether the applied bending moment exceeds the cracking resistance of the concrete section.

In its basic elastic form, the cracking moment is obtained from the flexural stress corresponding to the tensile strength of the concrete. PTKalc extends this expression using the transformed uncracked section and the additional effects described below.

$$M_{cr} = \frac{f_{ct} I_g}{y_t}$$

For the section-level calculation, PTKalc uses the transformed uncracked properties and includes the effects of shrinkage-induced reinforcement stress, axial compression and tendon eccentricity:

$$Z_{tr} = \frac{I_{uncr}}{y_t}$$

$$N_{total} = P_e + N_c$$

$$M_{cr} = Z_{tr}\left(f_{ct} - \sigma_{cs} + \frac{N_{total}}{A_g}\right) + P_e e_p$$

where:

- I_{uncr} is the transformed uncracked inertia;

- y_t is the distance from the transformed centroid to the tensile face;
- f_{ct} is the adopted concrete tensile strength;
- σ_{cs} is the shrinkage-induced reinforcement stress;
- P_e is the effective prestressing force;
- N_c is any additional compressive axial force;
- A_g is the gross concrete area;
- e_p is the tendon eccentricity relative to the gross concrete centroid.

The shrinkage-induced reinforcement stress is:

$$\sigma_{cs} = \frac{(2.5\rho_t - 0.8\rho_c)E_s\varepsilon_{cs}}{1 + 50\rho_t}$$

The calculated cracking moment is constrained to remain non-negative. This formulation means that prestress and axial compression increase the cracking resistance, while shrinkage reduces the tensile capacity available before cracking.

For normal reinforced concrete members, the tensile strength is determined in accordance with the adopted design standard.

For prestressed members, PTKalc incorporates the beneficial influence of prestressing by considering the reduction in tensile stress produced by the effective prestressing force before assessing the onset of cracking.

14.3. Cracked Section Analysis

For cracked sections, PTKalc performs a transformed section analysis.

The reinforcement is converted into an equivalent concrete area using the modular ratio

$$n = \frac{E_s}{E_c}$$

PTKalc first determines a creep-adjusted concrete modulus:

$$E_{c,LT} = \frac{E_c}{1 + \phi_{final}}$$

and then uses:

$$n = \frac{E_s}{E_{c,LT}}$$

The transformed uncracked section includes the concrete flange and web together with the net additional transformed reinforcement areas:

$$A_{t,tr} = (n - 1)A_t$$
$$A_{c,tr} = (n - 1)A_c$$

The transformed centroid is:

$$\bar{y}_{tr} = \frac{A_f y_f + A_w y_w + A_{t,tr} d + A_{c,tr} d'}{A_f + A_w + A_{t,tr} + A_{c,tr}}$$

and the uncracked transformed inertia is calculated by summing the local inertias and parallel-axis contributions of the flange, web and transformed reinforcement.

Accordingly, the reference inertia used in the local stiffness ratio is the transformed uncracked second moment of area, rather than the gross concrete inertia alone.

The neutral axis is then determined by satisfying force equilibrium within the transformed section.

Once the neutral axis has been established, the cracked second moment of area is calculated by summing the contribution of the concrete compression zone together with the transformed reinforcement using the parallel-axis theorem.

Unlike simplified design methods that assume one cracked inertia for an entire member, PTKalc performs the calculation independently at each finite-element end. The resulting end values are subsequently condensed to one effective stiffness for the element.

14.4. Effective Flexural Stiffness

The actual stiffness of reinforced concrete lies between the gross and fully cracked section properties.

PTKalc therefore determines an effective second moment of area that reflects the degree of cracking at each section.

The effective second moment of area is obtained from the implemented tension-stiffening relationship, which is governed by the ratio of cracking moment to the absolute service moment.

PTKalc first defines:

$$r_M = \min\left(\frac{M_{cr}}{|M_s|}, 1\right)$$

The effective inertia is then:

$$I_{eff} = \frac{I_{cr}}{1 - \left(1 - \frac{I_{cr}}{I_{uncr}}\right) r_M^2}$$

This is bounded by:

$$I_{cr} \leq I_{eff} \leq I_{eff,max}$$

The final upper limit applied by PTKalc depends on the member classification and reinforcement ratio:

Ieff <= 0.6 Iuncr

for prestressed sections, and also for reinforced concrete sections where:

$$\rho_{reo} \geq 0.005$$

For reinforced concrete sections below this reinforcement ratio:

Ieff <= 0.36 Iuncr

Accordingly, the final effective inertia is limited to 60% of the transformed uncracked inertia for prestressed and normally reinforced sections, and to 36% for lightly reinforced concrete sections. The lower bound remains the transformed cracked inertia.

The stiffness ratio passed to the finite element analysis is:

$$R_I = \frac{I_{eff}}{I_{uncr}}$$

This relationship provides a smooth transition between fully uncracked and fully cracked behaviour and reflects the tension stiffening contributed by the reinforcement between cracks.

14.5. Prestress Contribution

Prestressing significantly influences serviceability behaviour by reducing tensile stresses within the concrete and increasing the cracking resistance of the member.

The prestressing force contributes to the section response through

- axial compression,
- tendon eccentricity,
- global moments arising from the structural response to prestressing actions,
- reduced tensile stress,
- delayed cracking.

Accordingly, prestressed members generally retain a larger proportion of their gross stiffness than equivalent reinforced concrete members subjected to the same external loading.

The contribution of prestressing is evaluated directly during the section analysis and therefore influences both the calculated cracking moment and the resulting effective stiffness.

Prestress enters the local serviceability calculation in three principal ways.

First, the effective tendon force is converted to an equivalent reinforcement area:

$$A_{p,eq} = \frac{P_e}{f_{sy}}$$

This area is added to the conventional tension reinforcement before the combined steel area is transformed using the modular ratio:

$$A_t = \max\left(A_{s,reo} + A_{p,eq}, A_{s,min}\right)$$

For the serviceability transformed-section calculation, this minimum area is applied when establishing the effective tension-steel area for both reinforced and prestressed sections. This serviceability treatment is separate from the ultimate-design rule described in Section 3.8.

Second, the tendon acts as concentric compression in the cracking-moment calculation.

Third, its eccentricity contributes:

$$M_p = P_e e_p$$

to the cracking resistance.

The effective tendon force used by the wider analysis is calculated from the tendon arrangement as:

$$P_e = 0.85\, n_t n_s A_{strand} f_p$$

with the appropriate force-unit conversion.

The tendon depth is evaluated from the draped tendon profile at each finite element end, so the prestress contribution varies along the member.

14.6. Time-Dependent Behaviour

The stiffness of reinforced and prestressed concrete continues to change throughout the service life of a structure due to the time-dependent properties of concrete. The two principal mechanisms responsible for these changes are creep and shrinkage.

Concrete creep results in a gradual increase in deformation under sustained loading. Although the applied load may remain constant, the effective stiffness of the member decreases with time as the concrete undergoes additional strain. Shrinkage produces independent shortening of the concrete section and, where reinforcement or prestressing is present, generates additional internal stresses and curvature.

PTKalc incorporates these effects directly into the serviceability calculations by modifying the effective flexural stiffness of each finite element prior to the long-term structural analysis.

Unlike simplified methods that apply a single multiplier to the calculated deflection, PTKalc modifies the structural stiffness itself, allowing the redistribution of internal forces to occur naturally through the finite element analysis.

14.7. Creep Modification

Concrete creep is represented by reducing the effective flexural stiffness of each finite element in accordance with the adopted creep coefficient.

The final creep coefficient and the corresponding long-term element reduction are determined from the following relationships.

$$\phi_{final} = \phi_{basic} k_2 k_3 k_4 k_5 k_6$$

where:

$$\begin{aligned}
\alpha_2 &= 1 + 1.12e^{-0.008h} \\
k_2 &= \frac{\alpha_2 t^{0.8}}{t^{0.8} + 0.15h} \\
k_3 &= \frac{2.7}{1 + \log_{10}(t_{loading})} \\
k_4 &= \text{environmental factor} \\
\alpha_3 &= \frac{0.7}{k_4 \alpha_2}
\end{aligned}$$

For concrete strength not exceeding 50 MPa:

$$k_5 = 1$$

For higher-strength concrete, with the strength limited to 100 MPa:

$$k_5 = (2 - \alpha_3) - 0.02(1 - \alpha_3) f'_c$$

and currently:

$$k_6 = 1$$

The analysis duration is fixed at:

$$t = 365 \times 30 = 10\,950 \text{ days}$$

The final long-term reduction assigned to an element is:

$$R_{LT} = \frac{R_{I,ave}}{1 + \phi_{final}/0.76}$$

Therefore, the inertia used in the cracked long-term finite element model is:

$$I_{LT} = I_g \frac{R_{I,ave}}{1 + \phi_{final}/0.76}$$

The constant 0.76 is the age-adjustment factor adopted in the present PTKalc formulation.

Creep influences the analysis at two distinct levels. The creep-adjusted concrete modulus governs the relative transformed contributions of concrete and steel within the section calculation. The subsequent age-adjusted reduction represents the additional long-term reduction applied to the flexural inertia used in the global structural model.

The adopted coefficients and time-dependent relationships form part of the present PTKalc implementation and are applied consistently to reinforced and prestressed concrete members.

Unlike conventional approaches that apply creep only after structural analysis has been completed, PTKalc incorporates the reduced stiffness directly into the finite element model. Consequently, the redistribution of internal forces caused by creep is captured automatically.

14.8. Shrinkage Effects

PTKalc incorporates shrinkage into the section cracking assessment through the shrinkage-induced reinforcement stress. This stress reduces the tensile resistance available before cracking and therefore lowers the calculated cracking moment and effective inertia. Shrinkage consequently influences long-term deformation through the section stiffness assigned to the finite element model.

14.9. Element Stiffness Assignment

Once the effective long-term stiffness has been established, PTKalc assigns a unique flexural rigidity to every finite element within the structural model.

Each finite element is assigned a single flexural rigidity derived from the section results at its two ends.

At the start and finish of each finite element, the local effective-stiffness ratios are evaluated as follows:

$$\boldsymbol{R_{I,1} = \frac{I_{eff,1}}{I_{uncr,1}}}$$

$$\boldsymbol{R_{I,2} = \frac{I_{eff,2}}{I_{uncr,2}}}$$

The element effective-stiffness ratio is taken as the arithmetic mean of the two end values:

$$\boldsymbol{R_{I,ave} = \frac{R_{I,1} + R_{I,2}}{2}}$$

The long-term flexural rigidity is then:

$$\boldsymbol{(EI)_{LT,e} = EI_{g,e} \frac{R_{I,ave,e}}{1 + \phi_{final,e}/0.76}}$$

Regions subjected to high bending moments generally experience greater cracking and therefore possess lower flexural stiffness than lightly stressed regions. Since each element receives its own stiffness, the structural model naturally reflects these local variations in behaviour.

The result is a piecewise-varying stiffness distribution in which adjacent finite elements may be assigned different effective flexural rigidities, rather than the uniform stiffness assumed by many simplified beam analysis methods.

14.10. Long-Term Finite Element Analysis

Following completion of the serviceability calculations, the finite element model is reconstructed using the effective stiffness assigned to every element.

Separate gross and long-term global stiffness matrices are assembled from the corresponding element stiffness matrices:

$$[K]_{gross} = \sum_e [\,k_e(E, A, I_g)]$$

$$[K]_{LT} = \sum_e \left[k_e\left(E, A, I_g \frac{R_{I,ave}}{1 + \phi_{final}/0.76}\right)\right]$$

The same geometry, loads, connectivity, supports, end releases and axial areas are retained in both systems. Only the flexural inertia is reduced.

The two systems are solved as:

$$\{d\}_{gross} = [K]_{gross}^{-1}\{F\}$$
$$\{d\}_{LT} = [K]_{LT}^{-1}\{F\}$$

Since every element stiffness differs according to its cracking state and long-term behaviour, the global stiffness matrix differs from that obtained during the initial elastic analysis.

The resulting displacements therefore represent the structural response of the cracked, time-dependent structure rather than the original gross concrete section.

14.11. Force Redistribution

Modification of member stiffness alters the distribution of internal forces throughout the structure.

As cracking develops, the stiffness of heavily loaded regions decreases. During the second finite element analysis these regions attract smaller bending moments, while stiffer portions of the structure resist a correspondingly larger share of the applied loading.

Unlike simplified design methods that adjust member moments empirically, PTKalc captures this redistribution automatically through the finite element solution.

The redistribution is therefore compatible with structural equilibrium and member compatibility at every node.

PTKalc performs one explicit stiffness update. The effective section stiffnesses are calculated from the moments obtained in the initial elastic analysis, after which the structure is re-analysed using the reduced stiffnesses. The redistributed second-pass moments are recovered for the long-term response, but the section stiffnesses are not recalculated iteratively from those redistributed moments. The method is therefore a staged two-pass effective-stiffness analysis, rather than a fully converged nonlinear cracking analysis.

This procedure provides a more representative simulation of reinforced concrete behaviour, particularly for continuous beams where stiffness changes in one span influence the force distribution throughout adjacent spans.

14.12. Final Deflections

The nodal displacements obtained from the second finite element analysis represent the predicted long-term structural deformation.

These displacements include the combined influence of

- member cracking,
- reinforcement,
- prestressing,
- creep,
- shrinkage,
- structural continuity,
- stiffness redistribution.

Because the deflections are obtained directly from the reduced-stiffness finite element solution, compatibility is maintained throughout the structure and the long-term effects are incorporated through the adopted element stiffnesses rather than through a single global multiplier.

The resulting deformation profile provides the final prediction of long-term serviceability performance.

15. Summary

PTKalc integrates structural analysis, flexural design and serviceability assessment into a unified finite element framework. Rather than treating member design and structural analysis as separate processes, the calculated behaviour of each beam section directly influences the stiffness of the structural model.

This methodology enables the program to simulate the progressive effects of cracking, prestressing, creep and shrinkage while maintaining compatibility throughout the complete structure. By assigning effective stiffness values to individual finite elements rather than entire members, PTKalc captures the local variation in structural behaviour and provides realistic predictions of force redistribution and long-term deformation.

The adopted procedure is applicable to reinforced and prestressed concrete beams of rectangular, T-shaped and L-shaped geometry and has been developed to provide practical engineering solutions consistent with the requirements of Australian Standard AS3600

-end-.

Blank page