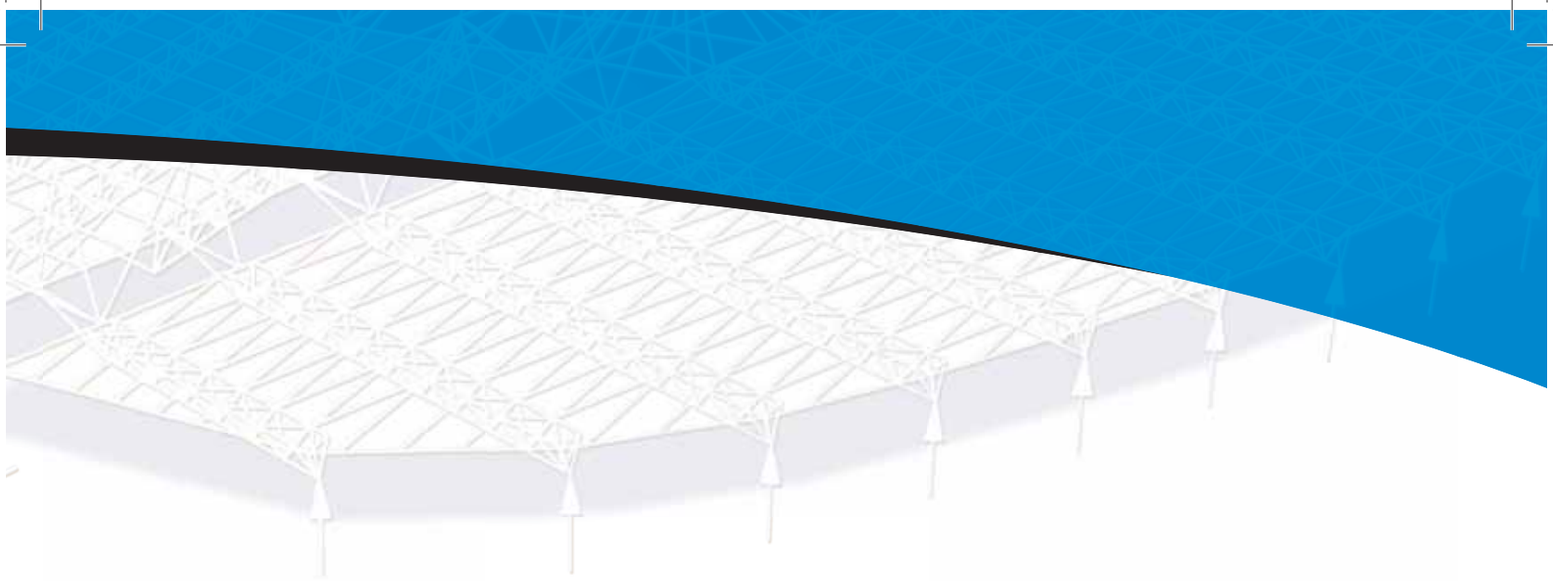


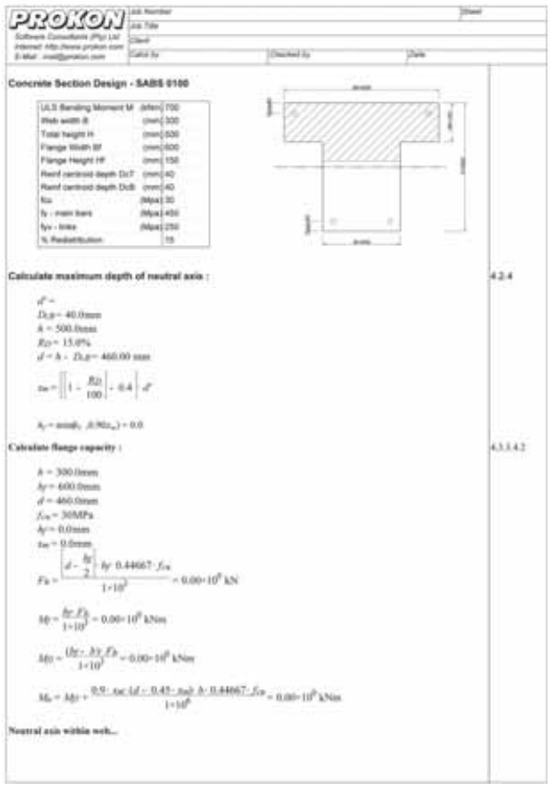
PRODUCT CATALOGUE

PROKON provides engineers with tools to streamline their workflow in the structural and geotechnical spheres. The tools are modular, but all are launched from the *Prokon Calcpad*, a very powerful object for interaction with the design modules. Export to PDF and a compact file format has made the *Prokon Calcpad* a winning tool for designers. All the modules boast an extensive help system, example files, support logging tools, and support by qualified engineers.

PROKON
Structural Analysis & Design

Calcpad

This is the main module from where you launch the various analysis and design modules. You can also use *Calcpad* to build calcsheets with design notes, drawings and equations. The user may customise the calcsheet layout to suit his/her style, using lines, text, labels, graphic files and adding header items. The *Prokon Calcpad* becomes the project database, containing all the output tables and graphs of the various modules, as well as imbedded input files.


PROKON

Concrete Section Design - SABS 0100

Calculate maximum depth of neutral axis :

Calculate flange capacity :

Neutral axis within web...

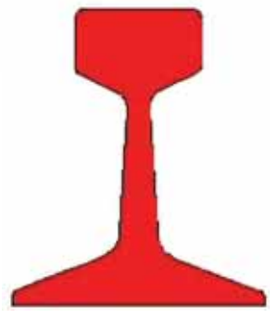
Live Update

The *Live Update* utility provides a platform for comparing and updating your Prokon installation. New versions can quickly be downloaded and will automatically be installed.

PROKON
Structural Analysis & Design

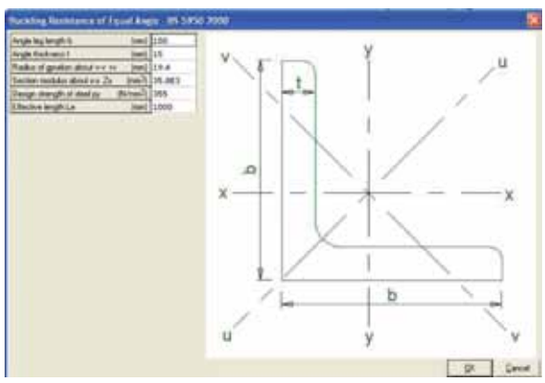
Section Database

The *Section Database* is a central repository for all sections used in the Prokon suite. This enables you to select sections by name in modules such as *Frame Analysis* and the section properties are imported directly from the database. The sections are divided into different materials, and then into different section shapes. The files differ per country.



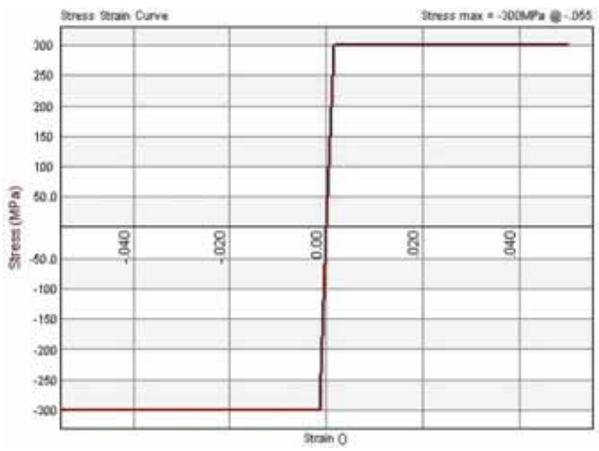
Scripting

The scripting tool provides a programmable interface to the *Prokon Calcpad*. The scripting toolbar contains extensive examples. Repetitive calculations can easily be automated by using the scripting feature and programmable equation objects. Scripts can control graphics, input grids, equations, calculations, reference text and much more. This enables the user to produce professional looking output for mundane tasks.



Material Database

The *Material Database* utility is a base component for all frame analysis modules. The database initially lists the structural properties of a number of typical building materials. These can be expanded to include more materials.



STRUCTURAL DESIGN MODULES

These can be divided into eight separate groups according to the structural material used, design method employed, or the specific application of the module.

STRUCTURAL ANALYSIS MODULES

Frame Analysis

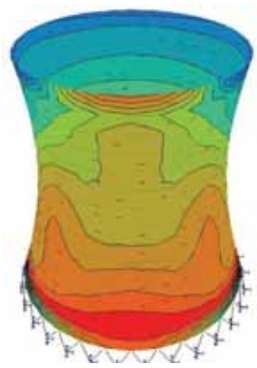
The *Frame Analysis* module supports the linear analysis of a two dimensional frame consisting of beam elements. This includes moment frames, trusses and/or grillage analyses. Temperature effects and prescribed displacements may also be added.

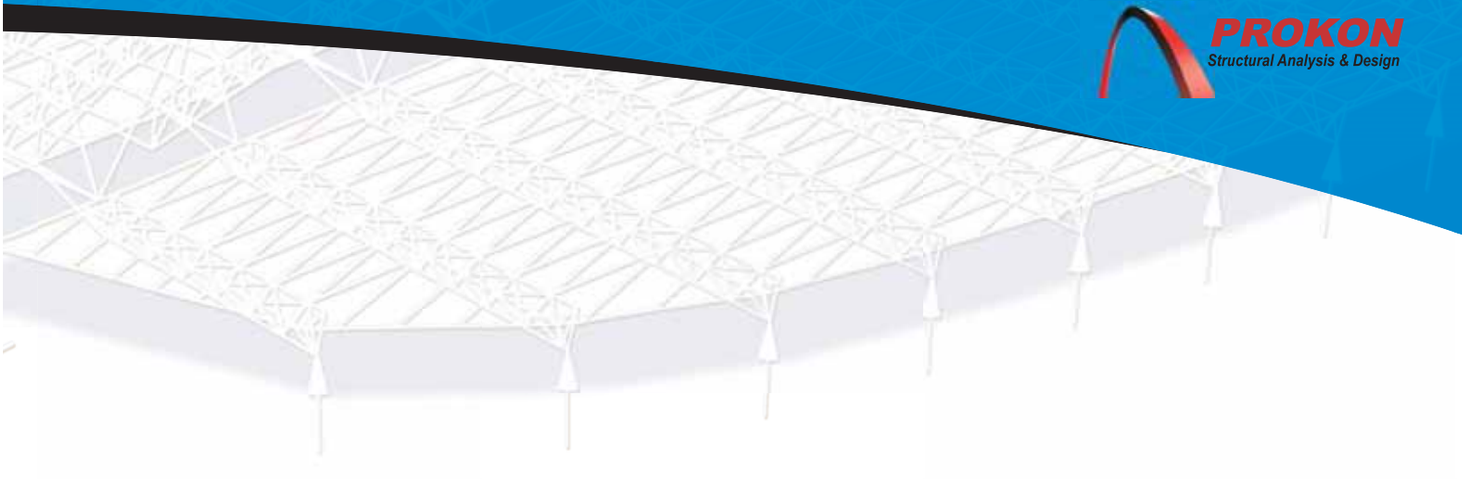
Displacements, beam end forces, moment envelopes, reactions at service and ultimate loads are given. The Frame module may be extended in several ways:

- *+3D extension:* Three dimensional analyses may be done, giving every node six degrees of freedom.
- *+Finite Element:* Adds shell, brick and catenary cable elements to the choice of input.
- *+Second Order & Buckling:* This allows the used to do a second order analysis on a frame, which allows certain savings in the steel design modules to be applied, or a buckling analysis, often used to point out structurally unstable frames.
- *+Non-linear extension:* Allows the user to analyse both geometric and material non-linear behaviour.
- *+Dynamic Analysis extension:* Calculation of mode shapes and natural frequencies as well as seismic and harmonic analyses are possible.
- Concrete design calculations can be done on the shell element output.

Frame analysis also links to design modules after an analysis has been done. In this way all load cases/combinations with the necessary forces are transferred to the applicable module. Output may be viewed and included in your calculations, in graphic format or in analysis reports.

Frame analysis also links to other software by way of .DXF, .DWG files, CIS 2 and .SNF files for steel detailing packages and by direct link to Autodesk Revit Structure.



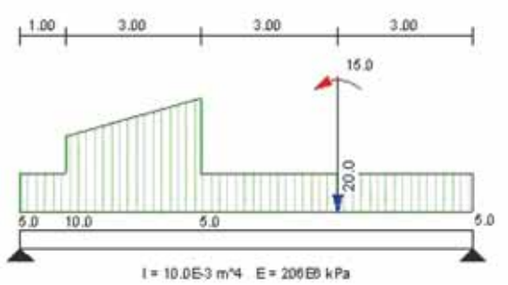


Plane Stress/Plane Strain Analysis

This module allows the user to enter any shape which will be meshed automatically and analysed using the method specified.

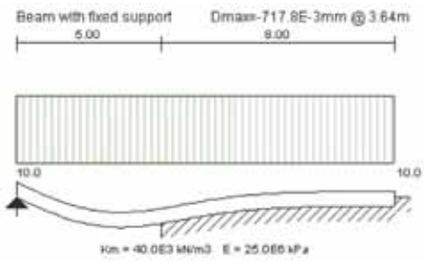
Single span beam analysis

This module is a basic analysis tool for beams – any end condition can be specified. Point loads, point moments and distributed loads may be entered for a complete evaluation of the member forces and reactions. The user may select beam sections from the *Section Database* or enter custom section properties.



Beam on elastic support analysis

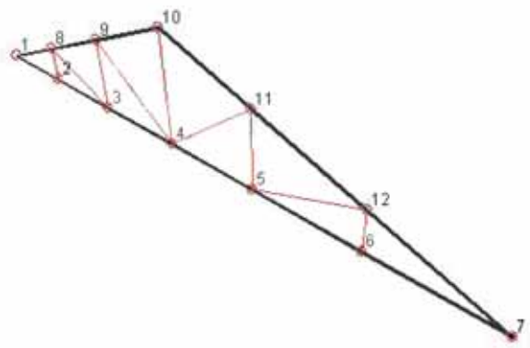
The user may enter multiple elastic supports, with or without gaps and any number of fixed supports. Diagrams of soil pressure, moment, shear and deflections are given as output.



STEEL MEMBER DESIGN MODULES

Member Design for Axial Stress

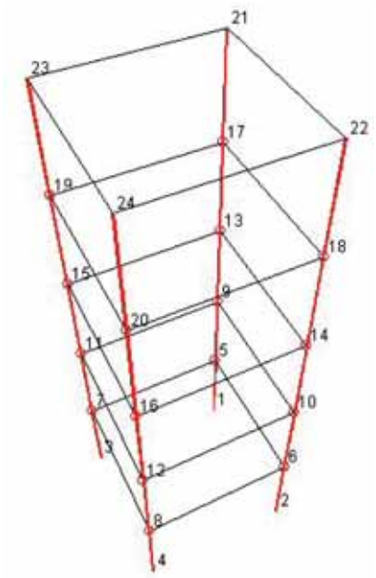
This module is used for checking and optimising steel members subjected to axial stress only, e.g. truss members. The input may be read from *Frame Analysis* output files or input may be entered on the interactive input page. All design parameters are editable e.g. maximum slenderness ratios per load case, effective length factors, etc. The design may be done to select the lightest possible sections, or to evaluate the currently selected sections.

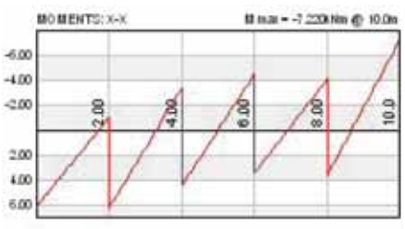


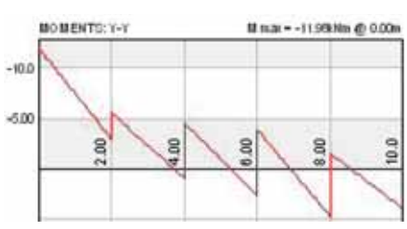
Member Design for Combined Stresses

This module is used for checking and optimising steel members subjected to a combination of axial force and uniaxial or biaxial bending moments, e.g. beams and columns in frames.

As with the *Axial Stress* design module, interactive input may be entered by the user, alternatively it also accepts input from the *Frame Analysis* and *Single Span Beam Analysis* modules. All design parameters are editable eg. maximum slenderness ratios per load case, effective lengths, etc. In addition to this, internal nodes can be defined and the accompanying effective lengths of each member for every axis may be adjusted. Individual output with all the necessary design equations may be added to the output file for each element analysed.



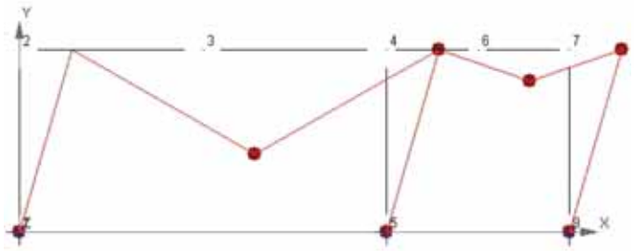




Plastic Frame Analysis and Design

This module does a linear or elasto-plastic analysis of a frame. The input is similar to that of the *Frame Analysis* program. All plastic hinges are shown graphically in the output. In all other respects the output is identical to that of *Frame Analysis*.

The use of plastic design methods is normally limited to the design of continuous beams and single storey frames with rigid joints, e.g. portal frames. It may also be acceptable to use plastic methods for designing some braced multi-storey planar frames.



Crane Gantry Girder Design

This module supports several classes of cranes, multiple steel sections, capping sections, multiple cranes on a single beam and more. It will quickly and accurately determine envelopes for all the required design forces, moments and deflections.

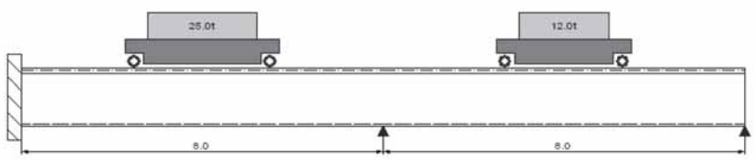
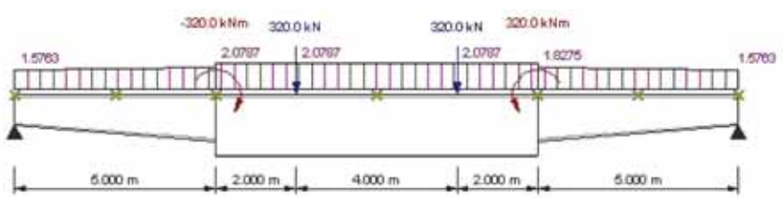


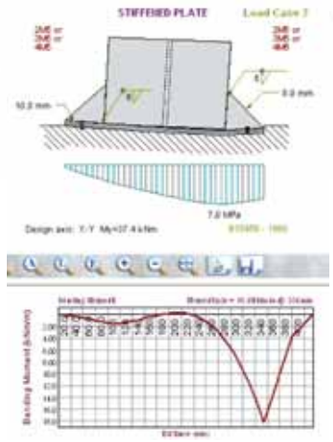
Plate Girder Design

This module supports almost any shape girder: stepped, straight or tapered girders may be entered with varying end conditions. Lateral restraints are indicated along the length of the beam. A full design report with calculations is generated, graphically indicating stiffeners, design forces, moments and deflections.



STEEL CONNECTION DESIGN MODULES

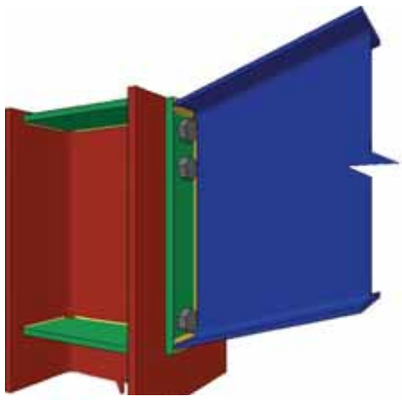
Base Plate Design

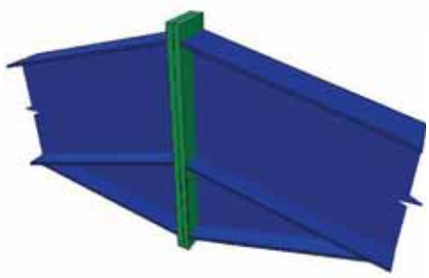


Base plates for all hollow sections and I or H sections may be designed. This module supports design for studs or bolts, stiffened or unstiffened plates, and produces graphic output of the bending moment in the plate, tensile forces in the bolts and the distribution of the compressive stress under the plate in the concrete substrate (for bolted option). It also produces a detailed drawing for manufacturing of the connection.

Beam to Column Connection Design, Apex Connection Design

All I and H sections (universal columns and universal beams) are supported. This is a moment connection design module and bolted or welded connections may be defined. The beam may be haunched and placed at any angle. There is an optimisation function which provides the user with a powerful tool to determine the best possible layout, eg. web plates, stiffeners, bolt sizes and spacing, etc. The design is immediately evaluated when any design change if made, and all design checks are marked as a success or failure. Full design calculations are given for all codes of practice. A manufacturing detail is provided after the design has been completed. Input may be transferred from *Frame Analysis* output through a design link.

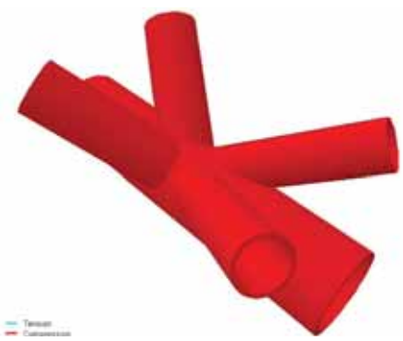




Hollow Section Connection Design

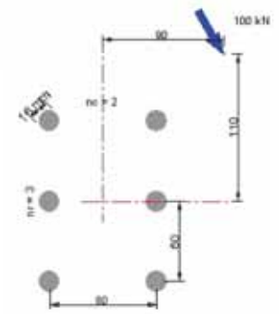
The *Hollow Section Connection Design* module does a complete design of welded structural hollow section connections. The connecting members may transmit axial force and can be circular, square or rectangular hollow sections. I-sections and H-sections (Universal Beams and Universal Columns) can also be used for the main chord.

Various connection layouts can be designed. These include K, T, N, X, and Y joints and combinations thereof.



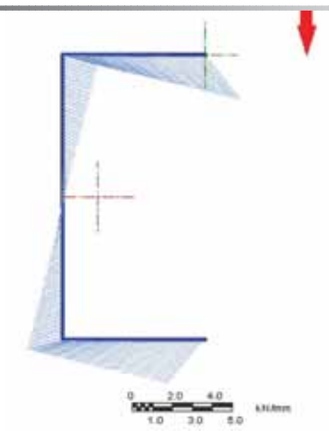
Bolt Group Design

This module evaluates eccentric forces on bolt groups, in bearing or friction grip connections, with the option for linear (Polar Moment) or non-linear (Instantaneous Centre of Gyration) analysis. All bolt forces are shown graphically and a minimum bolt size is determined.



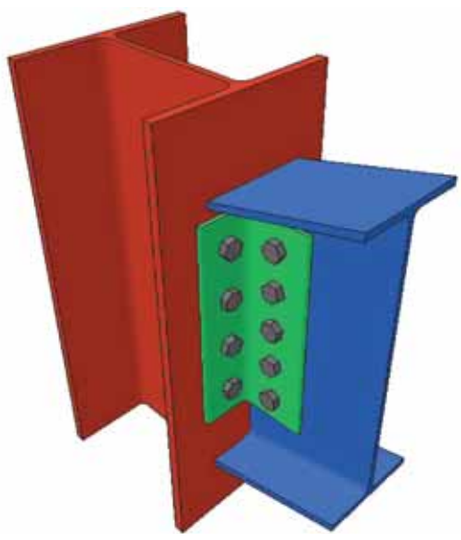
Weld Group Design

This module evaluates eccentric forces on weld groups, comprising fillet welds with the option for linear (Polar Moment) or non-linear (Instantaneous Centre of Gyration) analysis. The resistance of the weld is adapted for the force angle and the weld resistance is displayed graphically. An input wizard allows the user to place welds according to the most popular steel section outlines. A minimum weld size is determined.



Double Angle Cleat Design, Fin Plate Design, End Plate Design

These are simple connection design modules (no moment transfer). Sections may be chosen from the *Section Database*, and the connection elements may be optimised, as are the bolts and welds. Full calculations are shown with graphic output. 3D views, sections, elevations and plan views are shown with or without bolts and may be exported to CAD software as details.



CONCRETE DESIGN

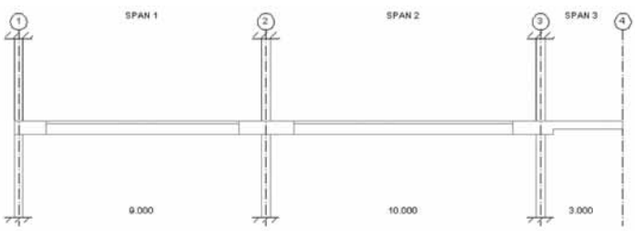
Continuous Beam & Slab Design

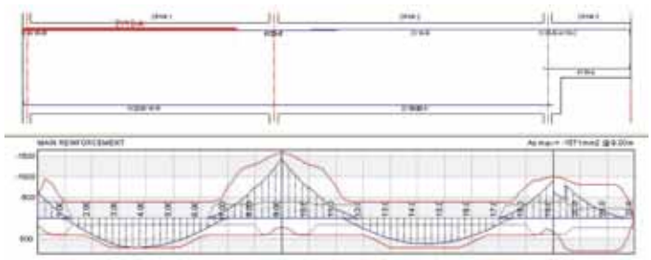
The *Continuous Beam and Slab Design* module is used to design and detail reinforced concrete beams and slabs as encountered in typical building projects. The design incorporates automated pattern loading and moment redistribution.

Cross-sections can include a mixture of rectangular, I, T and L-sections. Spans can have constant or tapered sections. Entered dead and live loads are automatically applied as pattern loads during the analysis. At ultimate limit state, moments and shears are redistributed to a user specified percentage. Both short-term and long-term deflections are calculated.

Complete bending schedules can be generated for editing and printing using *Padds*. The reinforcement details may be graphically edited by the designer, and is presented in user friendly pages depicting entered, required and minimum reinforcement (as specified by the applicable code of practice).

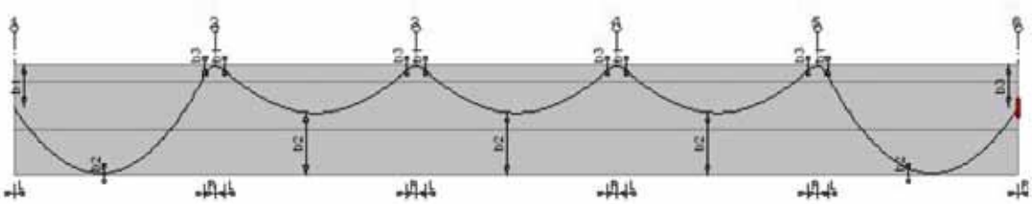
Frame Analysis output may be used to generate input for this module.





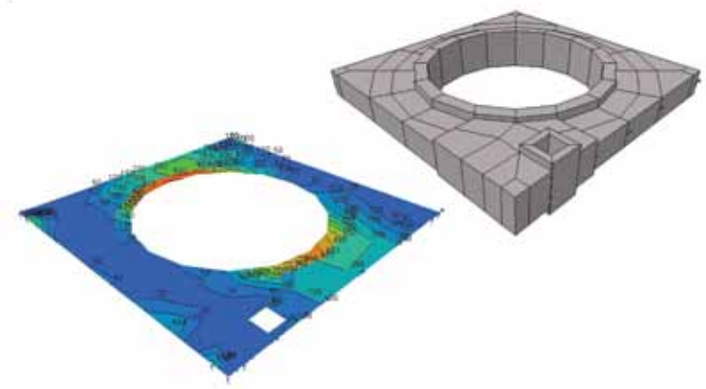
CAPTAIN : Prestressed Beam/Slab Design

CAPTAIN addresses all aspects of a post-tensioned system. Using graphic representations of all input, the designer can monitor and change all factors of the design at any time with immediate graphically understandable feedback. Functionality includes (but is in no way limited to) slab sections or user (eg bridge deck) sections, parabolic or harped tendons, automatic placement of tendons to balance specified loads, placement of additional rebar, punching shear design, profile scheduling, etc.



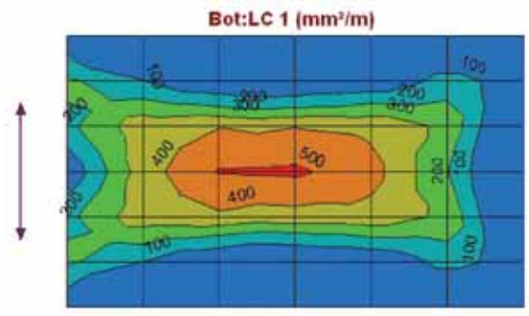
Finite Element Slab Design

Similar to the *Frame Analysis* module when using plate elements, input is generated to represent a slab. Input files for this module may also be generated with *Padds*. Point loads, and distributed loads may be added in load cases and then combined. Output is graphic and shows colour coded bands depicting contours of reinforcement values. Displacement, reactions, moments and reinforcement required (by means of Wood & Armer conversion) are also given.



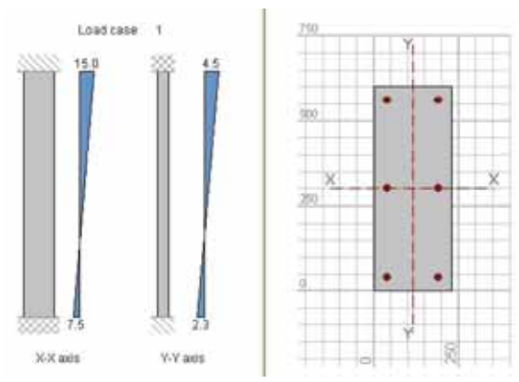
Rectangular Slab Panel Design

As indicated by the name, this module is used to design slabs by panel. The input requires the user to enter end conditions for every edge of the panel. The edge may be restrained for displacement (Support) and/or rotation (continuous). Point loads, line loads and UDL's maybe added in load cases with different load factors. Graphic contour output is given for moment, deflections and reinforcement. In addition to this an automatic detailing page will display a possible bending schedule and detailing layout. The user may change any of the reinforcement values on this schedule.



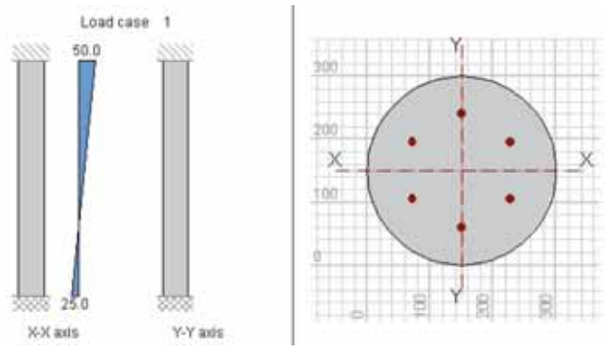
Rectangular Column Design

This is a design module for quick evaluation of a rectangular section. The required reinforcement is given, as well as a bending schedule in *Padds* format. The user is required to enter bracing and fixity parameters for both principal axes, and the column dimensions. Loads, including top and bottom moments about both axes, may be entered in load cases. The column is designed according to the specified code of practice and the interaction diagrams for both axes are plotted. The user can control the end conditions, stirrup layout and many more options when creating the bending schedule. *Frame Analysis* output may be used to generate input for this module.



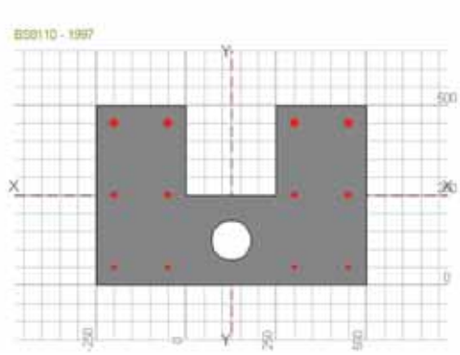
Circular Column Design

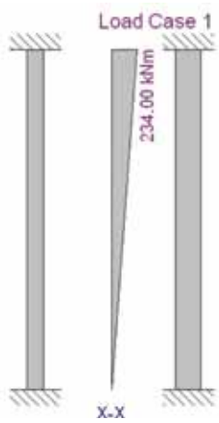
This is a design module for quick evaluation of a circular section. The required reinforcement is given, as well as a bending schedule in *Padds* format. The user is required to enter bracing and fixity parameters for both principal axes, and the column dimensions. Loads, including top and bottom moments about both axes, may be entered in load cases. The column is designed according to the specified code of practice and the interaction diagram is plotted. The user can control the end conditions, stirrup layout and many more options when creating the bending schedule. *Frame Analysis* output may be used to generate input for this module.



General Column Design

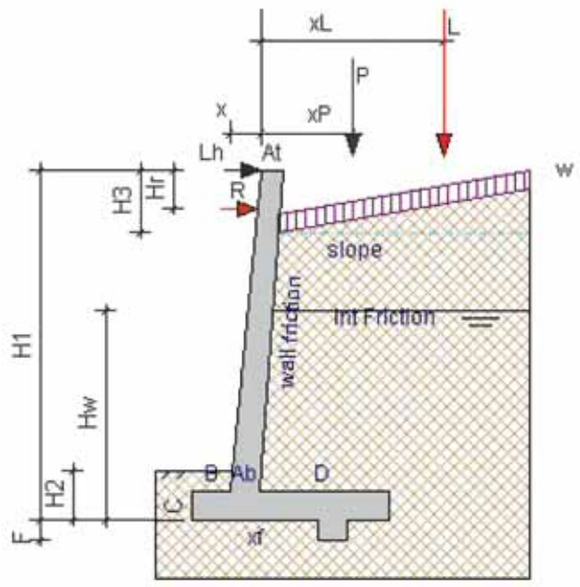
This is a design evaluation module for any general column section. No quick designs or reinforcement layouts are done. The user is required to enter bracing and fixity parameters for both principal axes, and the column dimensions. This may include any outline shape with or without voids in the column body. The user should also enter the positions and sizes of longitudinal reinforcement. Loads, including top and bottom moments about both axes, may be entered in load cases. The column is designed according to the specified code of practice and the interaction diagrams for both axes are plotted. Frame Analysis output may be used to generate input for this module, or Padds may be used to create the section of the column.





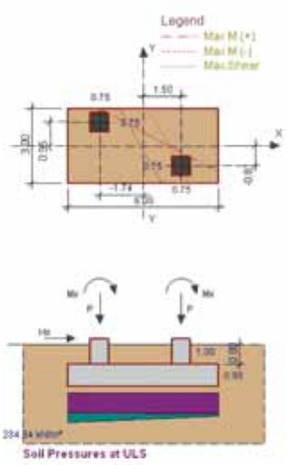
Concrete Retaining Wall Design

The *Retaining Wall Design* module is used to analyse retaining walls for soil and surcharge loads or seismic load conditions. Various types of walls can be considered, including cantilever, simply supported and propped cantilever walls. Complex wall geometry may be entered with many options for the design methodology. Rankine and Coulomb theory are supported and seepage may be allowed for if needed. Certain dimensions of the wall may be optimised. A full design is performed with safety factors for slip and overturning given at ULS and SLS. The moments in the walls are used to determine required reinforcement values. Al these values are shown graphically and an editable bending schedule is provided for a length of wall.



Concrete Base Design

Bases with columns, stub columns, or no concrete column may be designed. A maximum of two columns may be placed on the base, and diagrams of the stress distribution, with applicable safety factors and moments are provided during the design. A check for punching shear is also done. Finally a bending schedule with an exhaustive amount of options is provided. Input for this module may be derived from frame analysis via a design link.

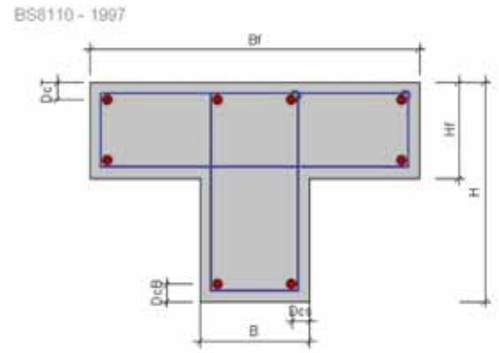


Crack Width Design

A section is analysed for crack width due to moment, direct tension and temperature loads. Several options for reinforcement size and spacing are provided with the accompanying crack widths, according to the design parameters.

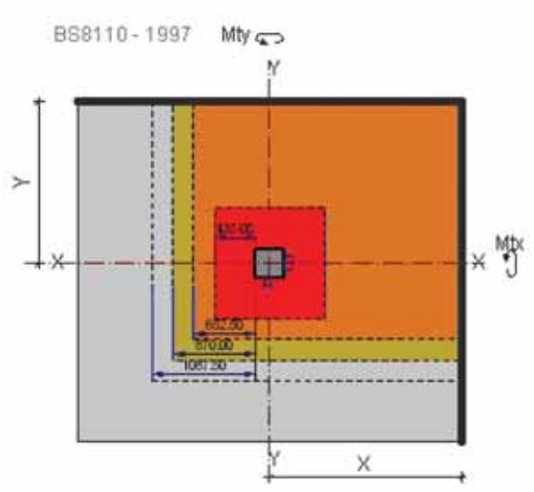
Concrete Section Design

The *Concrete Section Design* module is a simple utility for designing concrete sections for combined bending, shear and torsion. In addition to the forces normally accounted for in a continuous beam design, this module also accounts for, and reinforces for torsion moments. Full tabular design tables are provided.



Punching Shear Design

This module is designed for punching shear checks on reinforced slabs (for punching on post-tensioned slabs use *Captain*). The slab extremities relative to the column position need to be defined to determine whether they affect the perimeters. Longitudinal reinforcement in the two main directions may be specified as an area or bars at a certain spacing. Critical load cases with corresponding amounts of punching shear reinforcement are given as design values.

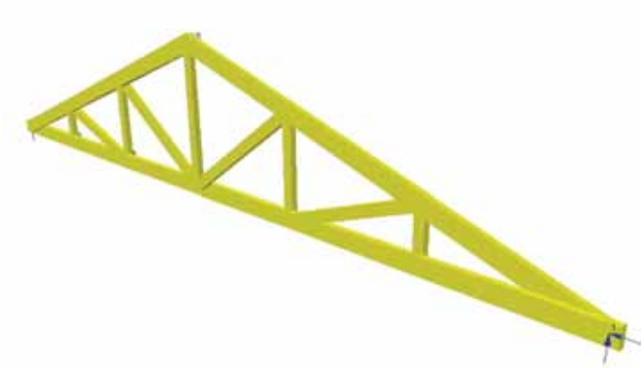


TIMBER DESIGN

Timber Member Design

The *Timber Member Design* module is used to check and optimise timber members subjected to a combination of axial and biaxial bending stresses, e.g. beams, frames and trusses. The program primarily acts as a post-processor for the *Frame Analysis* module. It also has an interactive mode for the quick design or checking of individual members without needing to perform a frame analysis.

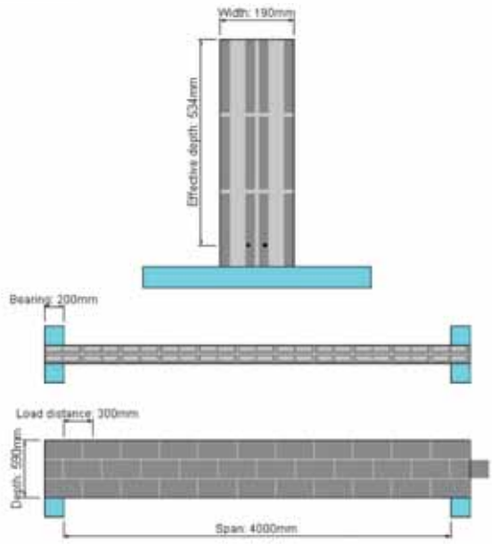
Timber and glued laminated timber load bearing members are supported.



MASONRY DESIGN

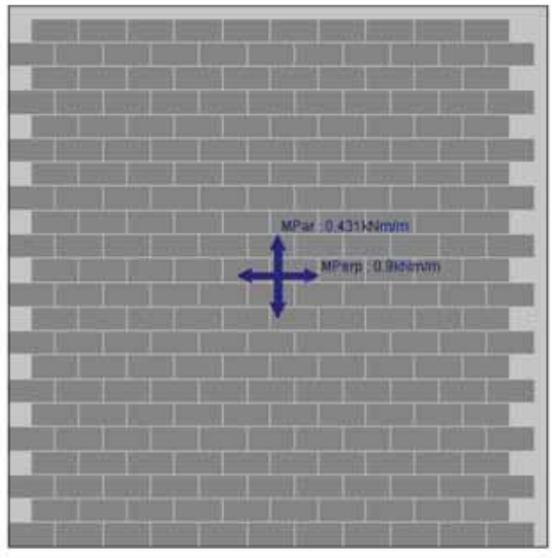
Masonry Section Design

Masonry Section Design is used for the design of masonry spanning openings. The module supports masonry units of arbitrary dimension and various nominal strengths. Characteristic masonry unit strengths can be specified or calculated based on the chosen code of practice. Single leaf, collar-jointed and grouted cavity arrangements are available. The output includes a design summary and sketch as well as a detailed report.



Masonry Wall Design

Masonry Wall Design supports the design of bearing walls (axially loaded) and wall panels (loaded normal to the plane of the wall). Similar to Masonry Section Design, masonry units of any dimension and strength can be used and the same arrangements are supported. The output also includes a design summary and detailed report.



CAD & DETAILING

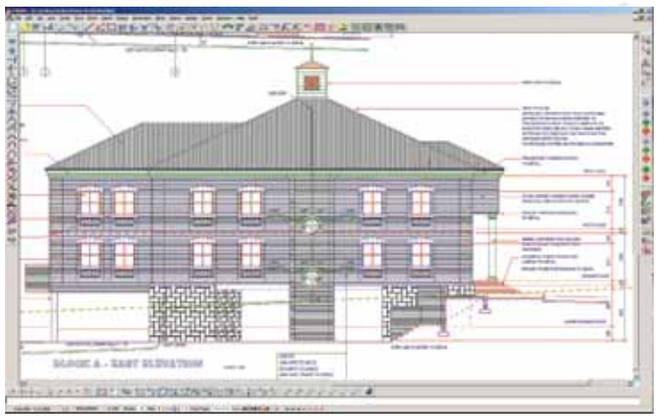
PADDS

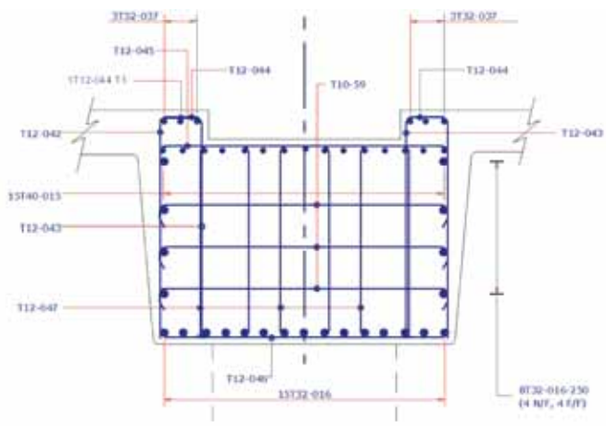
Padds is a CAD program designed specifically for the structural engineer. Al the basic CAD procedures are available to create or edit drawings. Elements like lines, arcs, circles, splines, text, hatch, dimensions, blocks, and construction lines are all supported with numerous creation and editing functions. In addition to this, *Padds* also links with the *Section Database* to enable users to draw sections, elevations and plans of all sections in the database. This greatly accelerates steel detailing It also contains a large number of reinforcement detailing functions and even contains functions for detailing complete staircases.

A full library of commonly used structural and architectural entities is available and the user can expand this repository at any time. Included in this is also a function to indicate welds with all supported symbols.

Scripting in *Padds* is one of its most powerful features. Users can create procedures to enter variable input, do calculations and create whatever their hearts desire in the *Padds* environment.

Another great tool is one for creating input geometries for other modules. *Padds* can create input files for *Frame Analysis, Finite Element Slab Design, Prosec and General Column Design.*





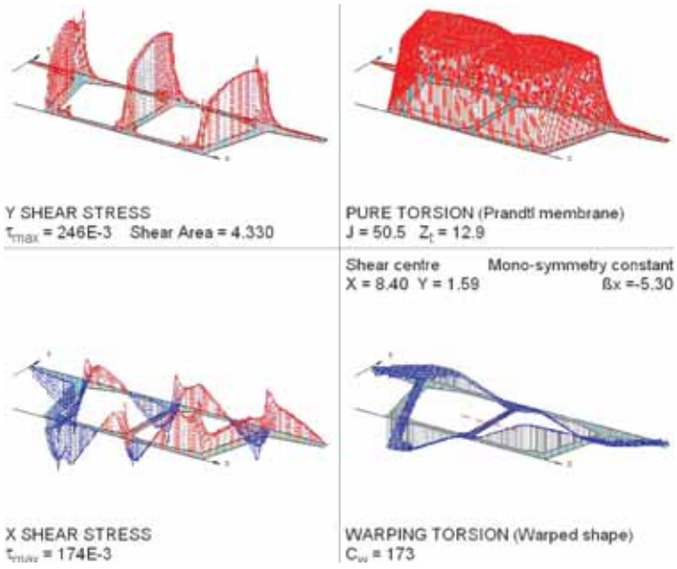
ProdoX

ProdoX is a tool for managing project documentation and information. It can handle outgoing and incoming documents. It uses a refreshingly friendly user interface with many automatic options. Given the automatic nature of the product, it is very customisable and the user has complete control over all options at all times.

GENERAL APPLICATIONS

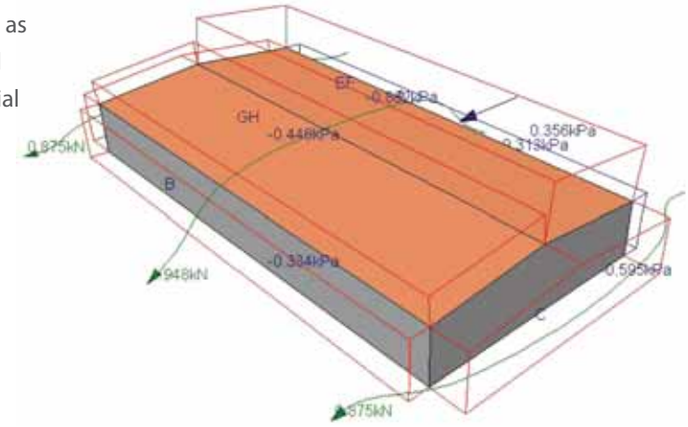
Prosec

Prosec calculates section properties for any conceivable single outline shape. The section may contain one or several openings. This product is used to create the section properties necessary for the *Section Database's* user sections. It computes the position and orientation of the major axes, all area properties, bending properties about major and minor axes, torsion properties and gives 3 dimensional renditions of the applicable stresses under these conditions.



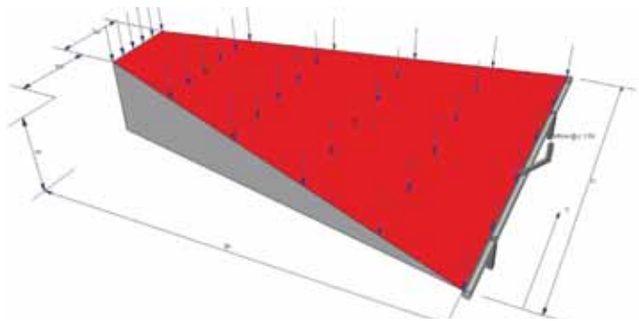
Wind Pressure Analysis

The *Wind Pressure Analysis* module is a simple utility for the calculation of free stream velocity pressures on building structures. It computes the wind pressures on the different pressure zones of various building geometries as recommended by SABS0160. It also computes frictional wind forces influenced by the specified cladding material and represents the vertical wind pressure profile to the eaves height of the specified structure. The input and output is represented with 3D graphics which gives the user a clear indication of wind direction and pressure vectors.



Gutter Size Design

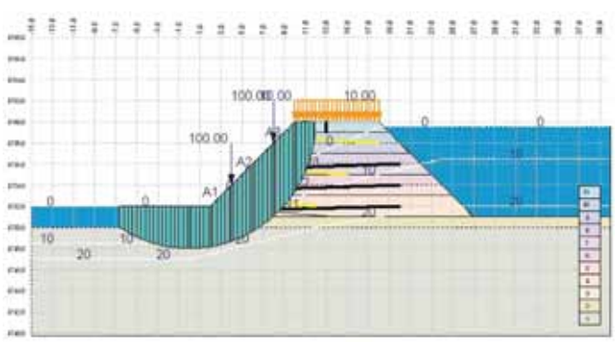
The *Gutter Design Module* is used to design gutters and down pipes to drain roofs of typical building structures for different rain intensities and durations. Input is displayed in a 3D visualization of the roof segment under analysis, and a roof draining and gutter draining diagram makes the interpretation of the result easy to understand. The outflow characteristics of each gutter segment and each down pipe is summarized in table form. Detailed equations of the computation are also provided.



GEOTECHNICAL DESIGN MODULES

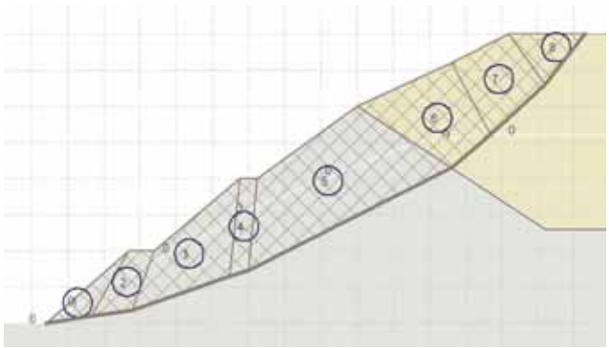
Generalised Slope Analysis

SLOPBG is a slope stability computer program which uses Bishop's Modified Method (1955) of analysis for the evaluation of the stability of generalized soil slopes. The slope may consist of materials with differing shear strength properties, defined either in terms of shear strength parameters or an un-drained shear strength profile. Water pressures, external loadings and reinforcement are included, to make the analysis as generalised as possible. The user is presented with a choice of searching for the critical minimum factor of safety circle or inputting a user defined circle. The deterministic analysis mode is supplemented by a probabilistic mode to evaluate the effect that the range of input values have on the FOS. The probability density function of the FOS is obtained using simulation techniques.



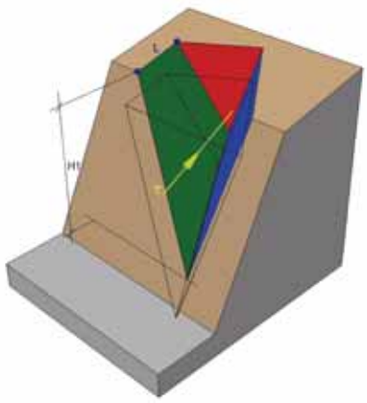
Generalised Non-circular Slip Analysis

SLOPNC, a generalized non-circular slope stability program, uses the non-vertical slice method as proposed by Sarma (1979), for the prediction of the factor of safety of general shape surfaces. As the boundaries are non-vertical, structural features such as faults or discontinuity planes may be included. Water pressures, external loadings and reinforcement are included, to make the analysis as generalised as possible. The user interface is very similar to Generalised Slope Analysis, and the module also gives the user the choice between deterministic and probabilistic analysis.



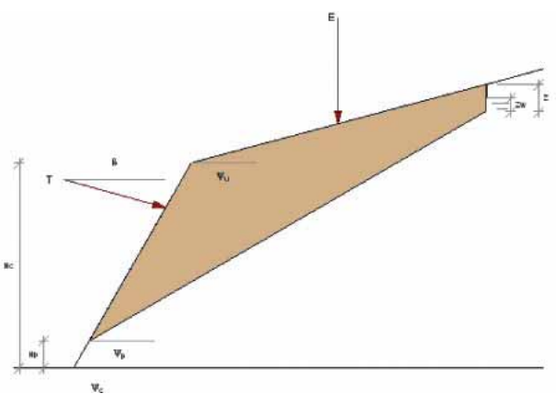
Tetrahedral Wedge Analysis

The program WEDGE determines the factor of safety (FOS) of a tetrahedral wedge that may form in a rock slope by the intersection of two planar discontinuities, the slope face, and the upper slope with or without a tension crack in the upper slope. The deterministic analysis mode is supplemented by a probabilistic mode to evaluate the effect that the range of input values have on the FOS. The probability density function of the FOS is obtained using simulation techniques.



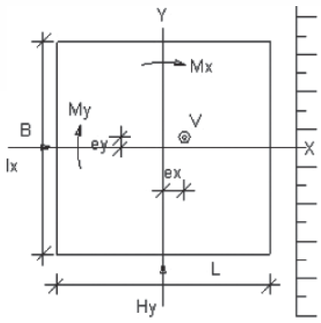
Planar Failure in Rock Slopes

The program ROCKPF determines the factor of safety (FOS) of a planar failure in rock. The deterministic analysis mode is supplemented by a probabilistic mode to evaluate the effect that the range of input values have on the FOS. The probability density function of the FOS is obtained using simulation techniques.



Bearing Capacity of Shallow Foundations

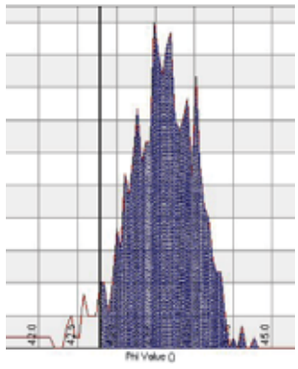
The program BCAP evaluates the ultimate bearing capacity of shallow foundations. The foundation may be circular, square or rectangular, the base and the external ground slope may be angled and the forces and moments imposed on the base may be from any direction. The drained and un-drained conditions are evaluated.



PLAN

Shear Strength of Jointed Rock Masses

The majority of rock masses, some granular soils and some dense sands, exhibit non-linear shear strength vs normal stress failure envelopes. The program ROCKJRM evaluates this non-linear shear strength envelope for a range of input parameters. Output may consist of instantaneous cohesion and friction values or the actual shear strength for a given normal stress. Deterministic and probabilistic modes are supported.



Shear Strength of Rough Joints in Rocks

The program ROCKJR evaluates the shear strength failure envelope for rough joints in rock. The analysis may be performed in deterministic or probabilistic mode. In probabilistic mode the range of input parameters are generated using simulation techniques to generate a probability density distribution of the output shear strength. The user is presented with the choice of two theories as advanced by Ladanyi and Archambault (1970) and Barton (1971a/b,73).