FEM-Design suite

Powerful and easy to use analysis and design from one central 3D model.

Utilise the power of FE for your designs of:
- Concrete slabs
- Shear walls
- Steel frames
- 3D shell structures &
- Complex 3D structures

FEM-Design is a group of modules that offers a powerful analysis and design environment using finite element methods.

Single elements or a complete building, made from any number of materials and structural elements can be analysed with ease. The users can create one central model to analyse their complete structure or focus solely on the analysis and design of specific horizontal or vertical planes or single elements.

From initial tendering stages to complete building analysis, FEM-Design offers a powerful, yet easy to use package that can be tailored to suite the exact analysis and design needs of the engineer and the job in hand.

Key features include:
- Intelligent data connections. Quickly and easily create your model from your architects’ layout by importing of IFC, DXF and DWG files.
- Analyse your applied design reinforcement including the ability to specify data for the cracked section analysis iteration.
- Quickly and accurately check the stability of your structure. Check designs for steel with automatic calculations of utilisation with critical members highlighted on-screen.
- Manage project changes easily. Create one quality report from within the program with interactive graphical windows to your design model and all results views.

FEM Design suite module summary:
- Plate, Wall, 3D Frame, 3D Structure.

Accessory modules:
- PreDesign, Documentation Editor, Section Editor.
FEM-Design overall package features

Modelling

• Simple and user friendly working environment with intuitive CAD tools for easy model creation and editing

• OpenGL for fast fly through rendered views, animated results and powerful 2D and 3D graphical representation of input geometry and results

• Intelligent data connections provide options to import DXF, DWG or IFC files for quick model creation

• Point-point, line-line and edge connections between structural objects for real life situation modelling

• Point, line and surface supports with rigid, free and/or elastic stiffness components

• Section editor for defining custom section shapes for bars, beams and columns with automatic property calculation

• Inbuilt structure wizard for quick and easy input of slab and frame systems

Analysis

• Various loading types including, point, line, surface, temperature, support motion and stress loads. Automatic wind and snow loading are also available

• Automatic loading combinations for unfavourable loads, combing permanent, variable and accidental loads

• Automatic optimal finite element mesh generation for any geometry with no element limit and the ability to mesh to load positions if required

• Options for advanced static, dynamic (eigen frequencies), stability and seismic analysis. FEM-Design can automatically convert load cases to masses for dynamic calculations and surface loads into line loads in frame structures

• Automatic solving of singularity problems in internal/design force results above supports

Design

• Choose your desired design code for reinforcement and steel from 10 national standards, including British, Danish, Eurocodes, Hungarian, Norwegian and Swedish.

• Steel design includes automatic utilisation checks for stresses, flexural buckling, LTB and web and flange buckling

• Choose your preferred results display from 3D graphs, contour lines, colour palette or sections for each display result

• Documentation editor, for quality multi-page documentation within the FEM-Design environment

• Input and output data can be generated in tabular form with the ability to use Microsoft Excel
FEM-Design plate offers a powerful and easy way to design concrete slab systems according to actual concrete behaviour. Features of the module include powerful cracked section analysis, punching calculations, shrinkage effects, and the ability to apply and analyse user defined reinforcement.

The basic outline model can be generated by the program's own CAD tools or imported via DXF/DWG or model based IFC links. The structural elements are defined with easy to use tools for plate regions, beams, column and wall supports. An inbuilt structure wizard is also available. This can be used to generate regular models including all structural elements and the applied loading.

The program's powerful mesh generator automatically produces a mesh adjusted to the current geometry, support and loading conditions. Designs can then be carried out according to EC2 and 9 other national codes with features such as cracked section analysis, shrinkage behaviour and punching shear calculations all taken into account.

There are several easy-to-use tools for defining the "accurate" static model: absolute and relative coordinates, user-coordinate system, drawing and editing tools, region operations, special point editors, object snap tools, layer system, etc. There is no geometrical limit and plates can be created with any shape and thickness values, beams/columns with any section and walls with variable thicknesses, walls with variable thickness, etc.

Further special features are also available, such as designs of slabs with varying thickness, modelling of orthotropic material behaviour and checking of plate corners for uplift above supports.

**Powerful reinforcement design**

An initial calculation provides required top and bottom bending reinforcement across the whole slab, together with the corresponding punching shear capacities, deflection results and crack widths. On the basis of the above results the user can define the desired (applied) reinforcement in any shape and direction.

The Växjö Library FEM-Design model images (left) show an example of the required reinforcement results. Areas in red highlight that not enough top reinforcement in the X direction has been provided.

The RC design module also suggests additional reinforcement required after taking punching shear into account. In addition to the numerical output, the columns are displayed in different colours depending
on the shear capacity with or without any additional shear reinforcement (e.g. green columns indicate that the shear capacity is OK with existing reinforcement, and red means that the shear capacity is too small even with additional shear reinforcement).

Cracked-section analysis
Deflections may be calculated using cracked-section analysis applied for serviceability load combinations and user-defined reinforcement. As well as deflection results, crack width and crack direction (crack lines) results can also be requested.

As an example, FEM-Design Plate module was used in the design of The Chimes Shopping Centre, Uxbridge, because of the great flexibility offered in placing future openings in the slab. The program also provided substantial savings in the reinforcement design compared with conventional methods.

Peak smoothing procedure
FEM-Design gives an automatic solution for addressing the singularity problem (peaks) of internal and design force results above supports. Automatic peak smoothing regions can be applied by the program in user specified locations (e.g. over columns, at beam and wall end points, etc.). These regions provide more realistic results and are calculated based on the column, beam or wall size and the thickness of the plate.

Geometry Import/Export
Two types of geometry import can be carried out in FEM-Design: DWG/DXF drawing and model-based IFC.

Design of ribbed slabs
In FEM-Design Plate, beams under the slab (i.e. ribbed slabs) can be modelled for static analysis including internal forces and displacement. The design results for design forces, areas of required rc and crack widths can all be displayed in various forms.

4 simple steps to concrete design:
1st step: Run the concrete design and analysis to obtain the required reinforcement and punching results.
2nd step: Define (applied) top/bottom reinforcement in the slab taking into consideration the previous design results.
3rd step: Run a check on the applied reinforcement combined with cracked-section analysis.
4th step: Analyse the deflection, crack width and missing reinforcement results and increase the reinforcement if needed. Repeat the checking process until the optimal reinforcement design is achieved.

FEM-Design Plate features
- Analyse and design slab systems with variable thickness and different materials
- Accurate modelling of orthotropic behaviour
- Uplift allowed above supports
- Wall and column supports
- Modelling of beams under slabs
- Dynamic analysis of vibration (eigen frequencies and vibration shapes)
- Concrete design calculations to several national codes.
- Consideration of shrinkage behaviour
- Shear capacity check
- Punching checking above columns
- Crack width/crack line distribution
- User-defined surface (applied) reinforcement as required reinforcement results
- Cracked section and displacement analysis of user defined applied reinforcement
- Required reinforcement in slabs (top and bottom) and beams (longitudinal bars and links)
- IFC connection for designing 3D structures by storeys (e.g. connection with ArchiCAD)
Within FEM-Design Wall, arbitrary 2D structures loaded in their own plane (stress in z-direction, $\sigma_z=0$) can be analysed for static and dynamic calculation and reinforcement design.

The Wall (plane stress) module offers design solutions for shear and load bearing walls, and also for other kinds of structural elements, such as webs of deep beams, steel plates, etc. Orthotropic conditions as well as regionally varying thickness can be defined.

The program is equipped with a powerful automatic mesh generator which generates an optimal finite element mesh for the current geometry, support and load conditions. With the Load groups tool (also available in other design modules), the program automatically generates all possible loading combinations including the most unfavourable from the defined load types (permanent, variable and accidental) and displays maximum and minimum result values. These features offer significant time saving as the all the loading combinations can be analysed in one operation.

Concrete walls are designed according to several national codes including Eurocodes. Design forces are then calculated as a combination of the normal and shear stresses in the wall. Tension reinforcement requirements are calculated and checks are made to ensure that allowable compressive stresses in the concrete are not exceeded.

A typical situation where the wall module can be used is in the analysis of a concrete wall with openings, used for stabilising a building against lateral loads, usually wind loads.

**Load-bearing walls**

The two results views of a wall design from the Hotel Westin, Warsaw (bottom left) show how the required reinforcement is concentrated in the lower part of the wall. Here the columns were used as structural elements in the lower storey to provide more flexibility, while concrete walls in the upper storeys were used to carry the load from the connected slabs.

**FEM-design Wall features**

- Wall-system with variable thickness and different materials
- Modelling of orthotropic behaviour
- Dynamic analysis of vibration (eigen frequencies and vibration shapes)
- Concrete design calculations to several nation codes
- Uplift allowed above supports
- Required tension reinforcement
- Checks that allowable compressive stresses within the concrete are not exceeded

**FEM-Design Plane Strain**

FEM-Design Plane Strain module can be used for structures extending widely in the z direction and where the loads, geometry and material conditions do not vary in that direction (strain in z-direction, $\varepsilon_z=0$). Examples of plane strain models are tunnels, dams, retaining walls and thick walled cylinders.
Advanced analysis of frame structures in 3D

Various calculation methods are available for 3D frame structures including: 2nd order analysis, dynamic analysis of eigen frequencies and vibration modes, seismic calculations according to Response Spectrum Analysis, stability and imperfection calculations, steel design and steel checking of bars having linearly variable sections.

Structures can be generated with the program’s own CAD tools or imported via DXF/DWG and model-based IFC links. A structure wizard is also available for creating common structure types.

A Section Editor is also available to create open or closed arbitrary shapes for bars, beams and columns used in the design modules. The program performs a very fast calculation of all parameters needed in 3D analysis (e.g. moment of inertia, shear factor, etc.) for the current section.

The results of calculations can be shown graphically as graphs in 3D for the whole structure, in userdefined 2D views, or for individual bars, beams and columns. Animation of displacement or vibration is also available.
Calculation of imperfection & global stability analysis
The program automatically calculates the probable imperfection shapes in real dimensions for steel bars according to EC3. This imperfection can be applied together with 2nd order theory. Without reference to material type, you can also run a global stability analysis for the whole structure and obtain buckling shapes with their critical load factors.

With “Soft spring” procedures, an automatic check of insufficient supports and connections can be run to solve structures defined as instable.

FEM-Design 3D Frame features
• Global stability analysis (“Subspace iteration”)
• Automatic process to calculate imperfections of steel bars according to EuroCode 3
• 2nd order analysis with consideration of imperfection shapes.
• Dynamic analysis of eigen frequencies and vibration modes
• Seismic analysis according to response spectrum analysis (modal analysis)
• Steel bar design: checks utilisation for stresses, flexural buckling, lateral torsional buckling, and for web and flange buckling
• Different type of connections between bar elements (rigid, hinged, semi-rigid, etc.)
• Automatic surface load conversion to line loads
• “Soft spring” procedure to solve structures having insufficient number of supports
• Curved beams
• Model import via IFC connection

Different connection types
The Point-point connection tool, allows different connections at the end points or intersections between connected or crossed bars to be set. With this feature, the real state of eccentric 3D bars or semi-rigid connections of columns and beams can be modelled.

Steel design
The steel design module enables you to check steel bars with regard to 1st or 2nd order theory (together with imperfections) for stresses, flexural buckling, lateral torsional and torsional buckling, for web buckling and flange buckling.
FEM-Design 3D Structure solves many construction problems containing mixtures of shell elements, slabs, walls, bars, beams and columns in arbitrary positions and loaded in any direction. Statical, dynamic, global stability, seismic analysis, concrete and/or steel design calculations can be run for the whole 3D-model.

The model geometry can be easily created in 3D with the help of several assistant tools such as axis and storey-systems, region operations, user-defined views and user-coordinate-systems. As before the user can also import DWG/DXF drawing files, 3D architectural or structural models and IFC model-based files.

For slab, wall and roof region elements, FEM-Design works with a common shell finite element with six degrees of freedom for each node. Bars, beams and columns have bar elements also with six degrees of freedom for each node. This makes it possible to analyse composite structures. There is no limit on the number of finite elements, and the program automatically generates the finite element mesh with the most optimal element sizes with automatic refinements at the supports and element connections.

The calculation results can be shown graphically as graphs, contour lines, colour palettes in 3D for the whole structure, in 2D by storeys (by slabs) or by vertical planes (by walls), or as arbitrarily placed sections by structural elements. Interesting numerical values can be shown in all images either by an automatic search and presentation for absolute values or local maximum and minimum values. In addition, the user can point with the mouse anywhere on the image to get a numeric result. To set the required views in 3D easily, dynamic zoom, pan and rotation commands are built into the program via powerful OpenGL technology.

As a special feature, solid definition and solid operation (union, subtract, intersection etc.) tools can be used to create arbitrary shapes and positions of shell elements.
Design of precast concrete elements
Enhanced edge and line-line connection tools allow connections between structural region objects with different displacement components (rigid, free, hinged or arbitrary connection stiffness) to be defined. For example, precast concrete elements can be easily modelled using shell elements.

Modelling and steel checking of variable bar cross-sections
Within the 3D Structure module, steel bars, beams and columns can also be analysed as shell elements, offering an accurate way of checking them for buckling (steel utilization). The “shell element” technology also allows the modelling of variable bar cross-sections, holes and stiffeners in webs, and sections built of different steel materials (non-homogenic steel sections).

Special load definitions
Several types of load can be defined in 3D structure, including (forces, moments, support displacement, stress and temperature loads). The program offers additional built-in features for loads: predefined load intensity values according to codes, fast algorithms to define surface loads, including varying intensity and automatic distribution of surface load onto bars (automatic conversion to line loads).
Connection among design modules & IFC model-based connection
FEM-Design allows seamless transfer between the different design modules. For example, a slab-system of a multi-storey building (defined in 3D Structure module) and its supports can be analysed in the Plate module for more detailed concrete design such as cracked-section analysis and column punching, etc. Load-bearing walls of a large 3D structure can also be designed in the Wall module. FEM-Design is also able to connect with other structural and architectural programs via the model-based IFC connection.

Documentation Editor
“Create your own document”
FEM-Design offers an easy and elegant way to create complete and multi-page documentation of projects containing a cover page with title table, chapters, figures, texts, tables, headers/footers, images and table of contents. Templates can be edited and stored to re-use standard styles for future new projects. Connections with other documentation programs (e.g. Microsoft® Word and Excel) are also available.

PreDesign
The perfect tender tool PreDesign can provide quick calculation results for 3D structures including reactions and internal forces of load-bearing walls and columns and provide estimated material quantities of the whole structure. Special tools such as copy storeys, display by storeys and automatic wind, snow and deviation load generation (as building information) ensure fast and easy definition of the complete statical model.

FEM-Design 3D Structure features
In addition to the features of FEM-Design 3D Frame 3D Structure offers the following:
• Steel bar checking for shell model including: buckling shapes and utilisation for flexural buckling, torsional buckling, and for lateral torsional and local buckling
• The ability to model straight bars with varying cross-section and bars having holes and stiffeners
• 3D modelling and analysis of structures containing a mixture of shell, slab, wall and bar elements