



**Finite Elements**

**Frameworks**

**Prestressing**

**Dynamics**

**BIM**





# Contents

Concept.....	1
Program Interface .....	2
Reinforced Concrete Construction .....	6
Prestressed Concrete Construction .....	10
Bridge Construction .....	12
Steel Construction.....	14
Dynamics .....	16
Nonlinear System Analysis.....	18
Fire scenarios .....	20
Timber Construction .....	21
Solid Models.....	22
Design Objects .....	24
NURBS .....	25
Data and Prices .....	26

# Concept

The **InfoCAD** program systems offer a number of software solutions that are specifically optimized for structural design and feature a range of tools for interactive modeling.

With the ability to calculate and check any structure, from simple slab systems and prestressed 3D shell structures to dynamically stressed solid models, the system can be used for a variety of civil engineering tasks. Model editing, analysis control and results output for all structure types are performed in a standard 3D CAD user interface. Special data interfaces ensure that

you have access to third-party programs. Thereby the exchange of digital building models (BIM) according to the IFC standard becomes more and more important.



## InfoGraph program systems - User-friendly high performance software for the growing demands of the construction industry.

### InfoCAD Options:

- 2D and 3D beam and shell structures, cable structures and solid models
- Laminate cross-sections, pile calculation
- Geometrically and physically nonlinear analysis
- Damage models for solid element constructions
- Soil Model according to Mohr-Coulomb, Phi-C-Reduction
- Spring elements with any nonlinear characteristics
- Determination of buckling eigenmodes
- Bending, shearing and torsional design, robustness reinforcement
- Punching shear check, fatigue check, concrete and steel stresses, crack width limitation
- European standards (Eurocode, BSI, DIN, OENORM and SIA)
- Design of Steel Structures for classes 1 to 4 (automatic classification, lateral torsional buckling)
- Design of Timber Structures per EN 1995-1-1 and EN 1992-1-2
- Dynamic calculations, modal response spectra method and pushover analysis for earthquake checks, time step integration, dynamic train crossing, nonlinear cable dynamics, dynamic collapse analysis
- Prestressing of beam, shell and solid structures
- Bridge checks according to EN 1992-2, OENORM, DIN Technical Report and Recalculation Guideline
- Computation of construction stages with creep redistribution
- Structural analysis for fire scenarios based on the general calculation method
- Design objects for stress integration at any section
- NURBS objects for handling free-form geometries
- Tetrahedron elements with contact properties
- Digital building models (BIM) acc. to the IFC Guidelines, BCF editor, BIM add-in for Autodesk® Revit®

# Program Interface



## Intuitive Operation

The **InfoCAD** program system has a graphical user interface where you can define entire structures along with their material and section properties and various loads. All common CAD functions such as positioning, copying, and mirroring are available for this purpose. The program also features layer and color management, which allows you to process even the most complex of structures. When developing the software we placed particular emphasis on making the interface both intuitive and user friendly. To this end we integrated a number of common Windows features

such as context-sensitive menus, detailed F1 help for all dialogs and so on.

## Easy-to-Use System Input

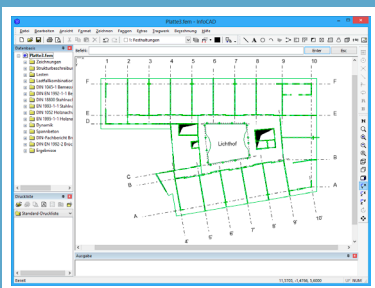
Structures are drawn and constructed either using the model objects found in the ground plans or as wire models. To complete and check the structures, the entered objects can be dimensioned automatically. DXF transfer can also be used as an alternate method of data entry.

## Automatic Mesh Generation

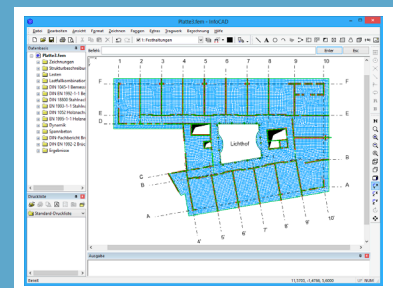
The FEM mesh can be generated automatically on the basis of the

existing construction. During this process all specified boundary conditions such as openings, down-stand beams, supports or support lines are taken into consideration.

Once the FEM mesh is generated, it can be modified with full user control over each individual element. Optimal modeling is supported by a number of sophisticated control functions. Additional semi-automatic mesh generators let you arrange elements in a way that satisfies all of your requirements.



You do not need to define surfaces or surface macros. The program recognizes all subareas and generates a mesh for them automatically.



## Material and Section

The standard steel and concrete classes as well as user-definable material types are available for all elements. The properties of these classes and types are used in all calculations and checks. Time-dependent variables for creep and shrinkage behavior are additionally available for the concrete classes. The  $\varphi(t,t_0)$  und  $\varepsilon(t,t_0)$  coefficients can be calculated as well, if required.

Area sections are described based on the element thickness. These descriptions can take orthotropy into account to allow for different rigidities for the main and lateral direction.

For beams and downstand beams, you can select parameterized standard sections as well as user-defined polygons. All common steel sections are made available in an extensive library.

## Loads

The numerous load types that are available provide a convenient means to implement all conceivable stresses. They are applied directly to the system in the graphical user interface and can be positioned anywhere you want. The load types include:

- dead loads
- single, line and area loads
- support displacement
- creep and shrinkage
- temperature
- hydrostatic pressure
- influence lines and surfaces
- prestressing

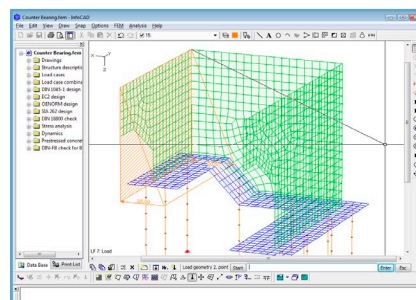
A special function can be used to generate moving loads as well.

## Actions and Design Situations

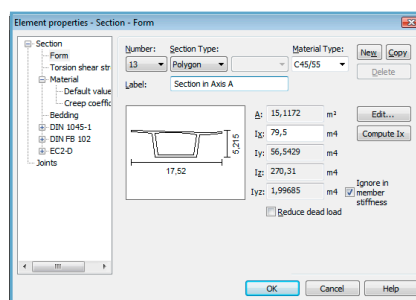
The design values of the load are calculated automatically based on the internal forces of the load cases and load case combinations.

By using cyclic permutation, the program constructs all variations of loadings actions and accompanying actions with regard to the partial safety factors and combination coefficients of the relevant standard.

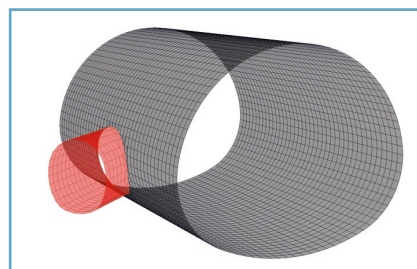
The extremal internal forces then provide the determinant design values for the respective situations. The load cases involved in a result and their respective weightings can be determined at a click of the mouse.



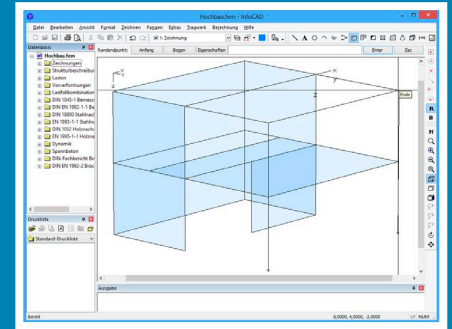
Load input at the system



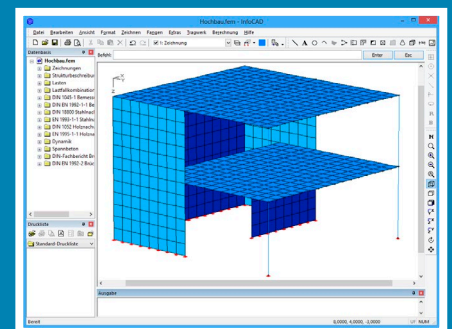
Definition of element properties



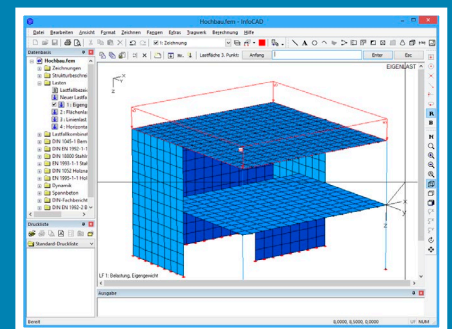
Automatic mesh generation at a complex tunnel connection



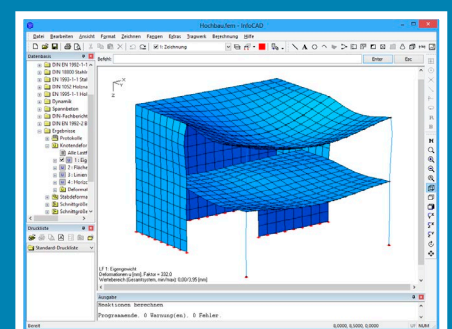
1. Draw ground plan



2. Generate mesh

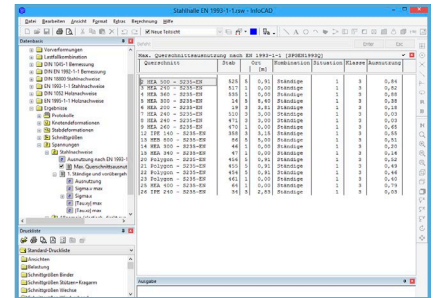
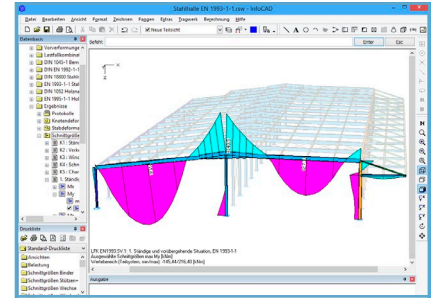
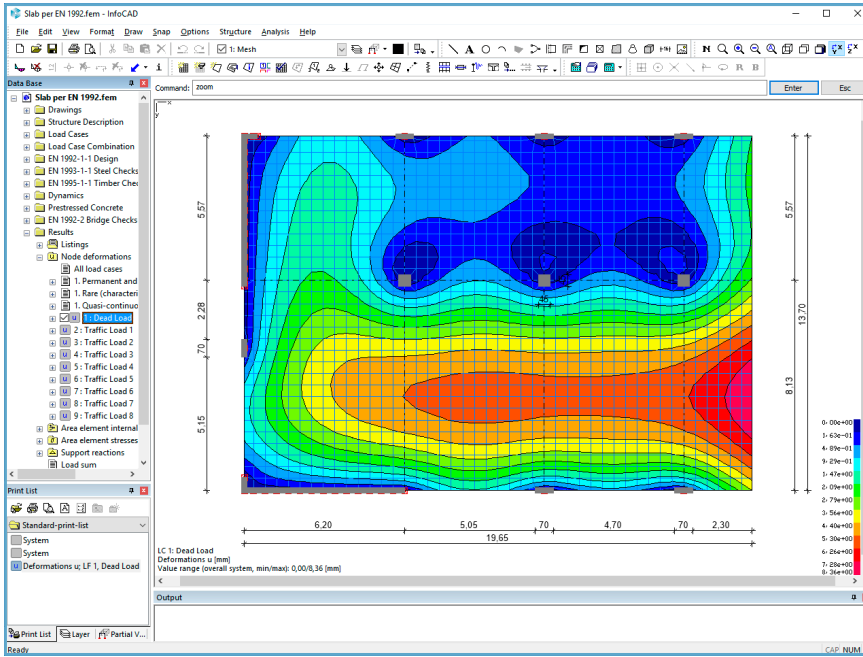


3. Apply loads



4. Access results

# Program Interface



*Stresses and utilizations*

## Results

All calculated results are shown in a clearly arranged tree structure. The available view options can be chosen depending on the current result. In addition to the various graphical views, all of the results can also be plotted numerically. A variety of setting options allow you to fine-tune the presentation. All checking programs generate additional custom logs that are configurable. They provide an easy way of understanding the complete computation.

## Individual Results

All available system data and results can be viewed in compact form by simply clicking an element. This offers you a simple and detailed method for checking data.

## System Viewer

The InfoGraph System Viewer offers a descriptive visualization of structure sections. It is especially well-suited for presenting and animating static or dynamic results. The views can easily be exported as individual images.

The realistic system view gives you an additional means of checking the computer model.

## Tabular Display

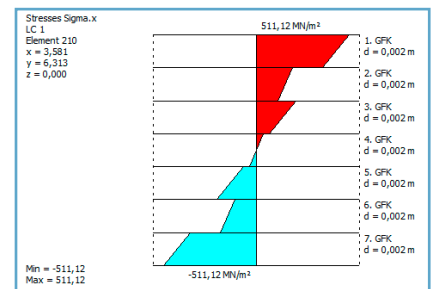
With respect to beam structures, it is often desirable to display the results as a table. You can switch between the graphical and tabular view with a simple click of the mouse. The table data can be exported to any Windows program via the clipboard and then processed further if necessary.

## Print List

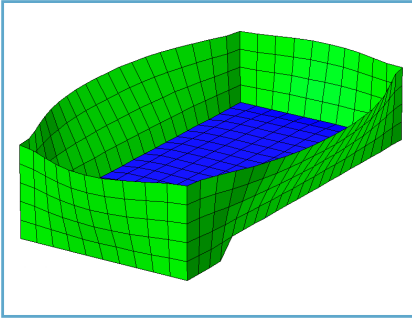
All the system data you want to print out at a later time can be stored in a print list at the push of a button. This option is available for both tabular and graphical views. You can store, for example, system views, section data, design specifications and results displays in any order you want. The print list can be expanded into a complete system analysis with its own sections and an automatically generated table of contents. Also, the data is always updated after system changes.

## Printout

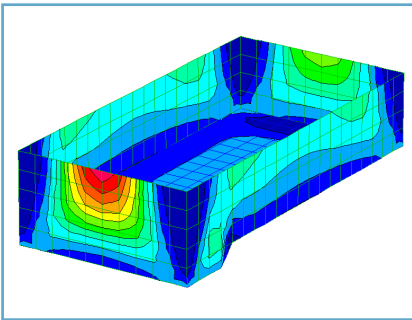
The current processing status can be printed out at any time. You can use the Windows Print Preview option to check the printout in advance. Individual graphics, tables or the entire print list can be printed. All of the printing devices recognized by the operating system are available.



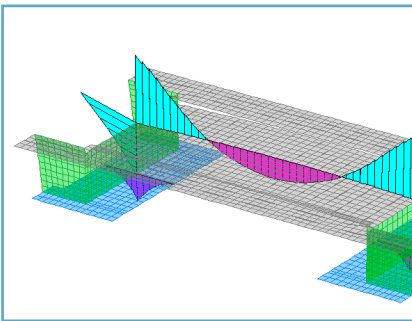
*Stress distribution in a laminate element*



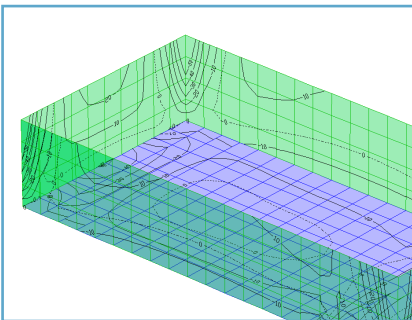
Deformation shape



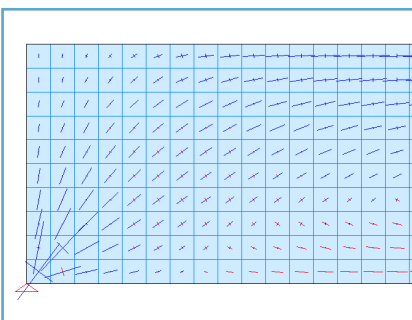
Color gradient areas



Section view



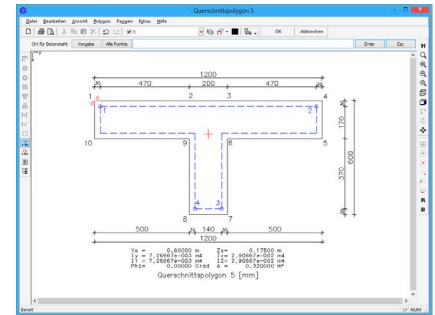
Contours



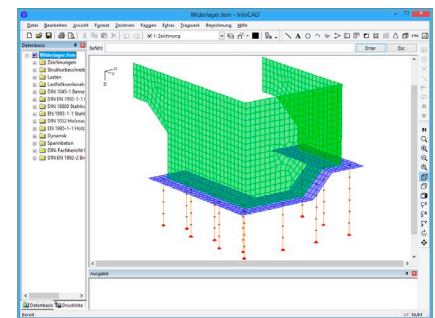
Vector view



# Reinforced Concrete Construction



Design section with steel layers



For users of a software program for solid construction, the implementation and integration of the relevant design standards and the comprehensibility of the results are of critical importance.

For this, **InfoCAD** provides comprehensive functions such as the single design or detailed listings for each element. The latest eurocode standards with the national annexes and the swiss standard are supported.

## Beams, Columns, Downstand Beams

The concrete section and its steel layers are entered in the section dialog. The typical standard sections used in concrete construction are available as predefined templates. You can also import complex geometries via DXF.

The bending design is carried out for two-axes bending with normal force at this polygon section, which can have any shape. For shear and torsion design, the existing equivalent sections are used.

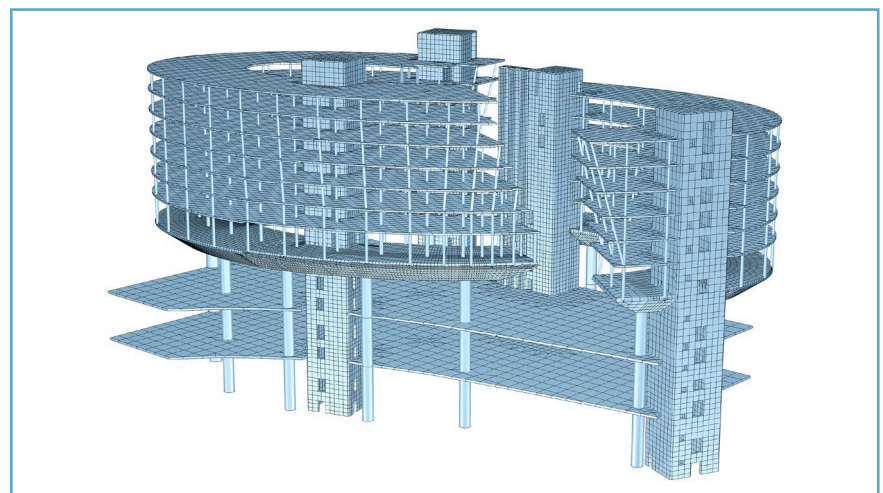
## Area Elements

Slabs, plain stress elements or shells are described by their section thickness. The edge clearances of the individual reinforcing steel layers can be controlled separately for each direction and specify the effective height for the lateral force design.

The design internal forces for the longitudinal reinforcement are determined using the plasticity approach from Wolfensberger and Thürlimann, which takes into account how much the reinforcement deviates from the crack direction.

Slabs with inclined reinforcement assemblies are designed according to Kuyt/Rüsch, in which the design moments are calculated with the help of principle moments  $m_1$ ,  $m_2$ . For a combined load, the normal design forces are determined from  $n_1$ ,  $n_2$  and are taken together with the design moments as the basis of the calculation.

The shear design is performed for the extremal lateral force of the design  $q_r = \sqrt{(q_x^2 + q_y^2)}$ .





## Actions and Combinations

Based on actions, the program uses the relevant safety and combination coefficients to automatically construct the design situation for the ultimate limit state and the serviceability limit state. The resulting extremal design values are then provided for the checks. Alternatively, you can also carry out the checks for all combinations rather than just the extreme values of the load.

## Ensuring Ductile Component Behavior

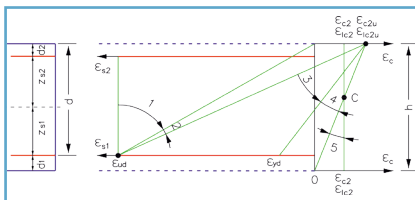
To prevent a component from failing without notice during initial crack formation, a minimum reinforcement is put into place to cover the crack moment with

$$A_s = M_{r,ep} / (f_{yk} \cdot z_s).$$

## Bending Design

As a part of bending design, the load-bearing check is performed for bending with or without longitudinal force and longitudinal force only for spatially stressed beam and area sections.

The reinforcement required for each internal force combination is determined with regard to the limit strain curve specified by the relevant standard. The final result is derived from the extreme value of all calculated reinforcements. When performing a compression member design, the reinforcement is symmetrically arranged and given the appropriate minimum reinforcement.



Strain ranges for the bending design

## Lateral Force

Lateral force design involves determining the diagonal tensile reinforcement and includes a concrete strut check. The necessity of a lateral force reinforcement is analyzed first.

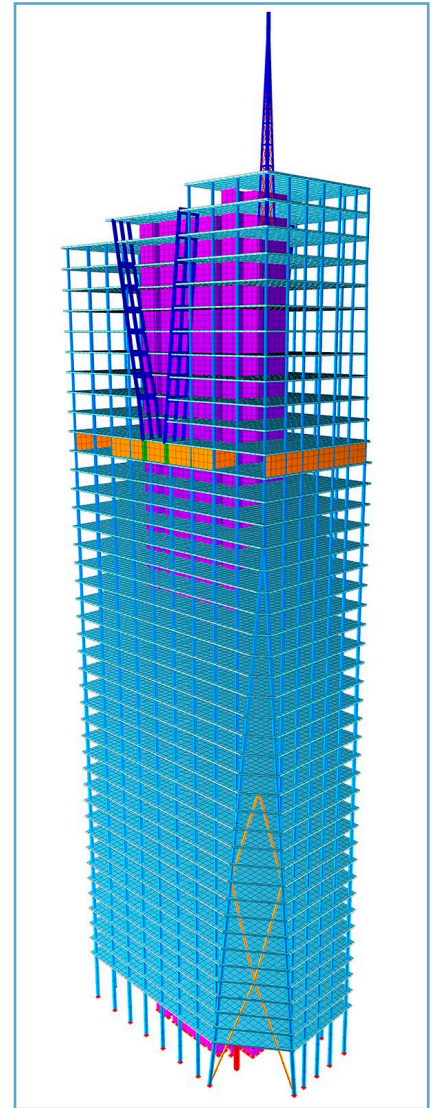
For components with the required lateral force reinforcement, the design value of the concrete longitudinal stress is taken into account and the strut angle is limited depending on the amount of stress.

The necessary minimum reinforcement is shown.

Optionally, you can also raise the allowable bending tensile reinforcement to prevent shear stirrups.

## Torsion

Torsion design involves determining the diagonal tensile reinforcement and the longitudinal reinforcement and includes a concrete strut check under maximum torsional stress, combined with a concrete strut check under lateral force.



## Checks for reinforced concrete construction:

### Checks at the ultimate limit state

- Minimum reinforcement for ensuring ductile component behavior
- Bending with or without normal force or normal force only
- Lateral force under consideration of the min. level of reinforcement
- Pure torsion and torsion with lateral force
- Check of the shear joint
- Checks against punching shear
- Checks against fatigue (concrete and reinforcement)

### Checks at the serviceability limit state

- Limiting the concrete compressive stresses
- Limiting the reinforcing steel stresses
- Minimum reinforcement for the crack width limitation
- Limiting the crack width via direct calculation
- Limiting deformations

# Reinforced Concrete Construction



## Punching Shear

The load-bearing safety check with respect to punching shear is performed interactively at the support of a slab structure. During this check, the determinant punching force can be taken from the static calculation (e.g., from the permanent and temporary or accidental design situation).

The program makes recommendations for the required longitudinal and/or stirrup reinforcement. The location of the supports in relation to the edge and any openings are taken into account. All necessary perimeters as well as the minimum reinforcement are checked and then applied. A detailed log documents the entire calculation.

## Fatigue Check

The fatigue check for concrete and steel is carried out for load-bearing components that are not subject to primarily static actions.

The stress range originating in condition II is checked for each steel layer and increased if necessary. The check for concrete under pressure is likewise carried out at the cracked section.

## Limiting the Concrete Compressive Stresses

Concrete compressive stresses are limited to prevent longitudinal cracks from occurring under the rare action combination.

If the serviceability, load-bearing capacity or durability of the structure is predominantly influenced by creepage, then this also need to be limited under the quasi-continuous action combination.

## Limiting the Reinforcing Steel Stresses

The steel stresses are checked by determining the strain state at the cracked concrete section. The tensile stresses are limited under the rare action combination for this check. The reinforcement corresponds to the maximum value from the robustness, crack and bending reinforcement, including a

possible increase as a result of the fatigue check.

## Crack Width Limitation

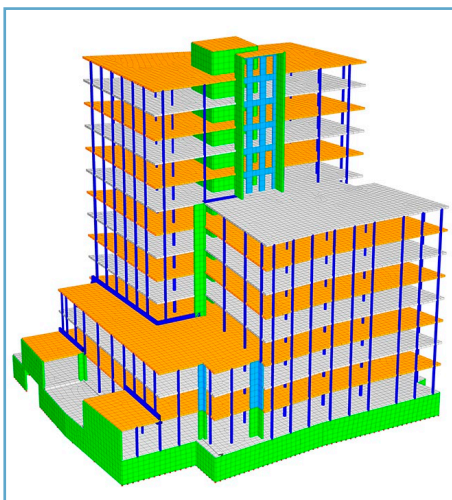
Crack formation is essentially unavoidable in areas where concrete is subject to tensile stress. The crack width should be limited such that proper use of the structure as well as its appearance and durability are not compromised as a result of cracks. The crack width limitation check includes checking the minimum reinforcement and calculating the crack width.

## Minimum Reinforcement for the Crack Width Limitation

This reinforcement is dimensioned to compensate forced actions and residual stresses and designed for the internal force combination that leads to initial crack formation in accordance with the corresponding requirement class.

## Crack Width Check

The crack width is determined for the final longitudinal reinforcement (maximum from the robustness, crack and bending reinforcement including a possible increase resulting from the fatigue check). The steel stress of the reinforcement is derived using the check combination defined by the requirement class



in condition II.

The reinforcement is increased until the prescribed crack width is maintained.

### Limiting Deformations

The deformations in the serviceability state are determined with regard to the existing reinforcement and the bending rigidities that result from it.

A residual tensile stress can be used to record how the concrete contributes to the tension between the cracks (tension stiffening).

### Determining Maximum Reinforcement

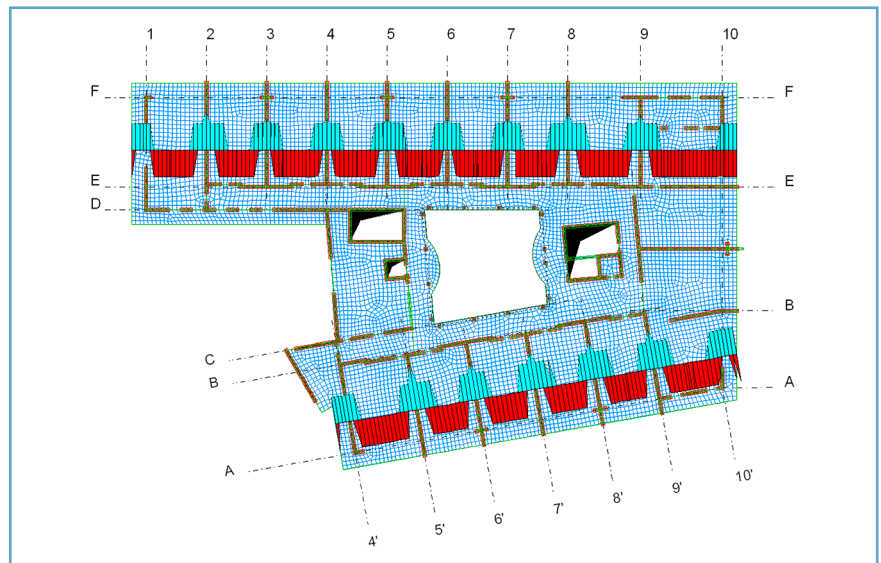
The design of the extremal internal forces from the action combinations does not necessarily lead to the maximum reinforcing steel reinforcement. Frequently there are combinations with lower internal forces that result in higher reinforcement levels.

Therefore, all internal force combinations can be checked in the InfoCAD program system as an option.

### Load-Bearing Safety Check and Deformations in Condition II

The nonlinear system analysis for all standards and structure types can be used to perform a load-

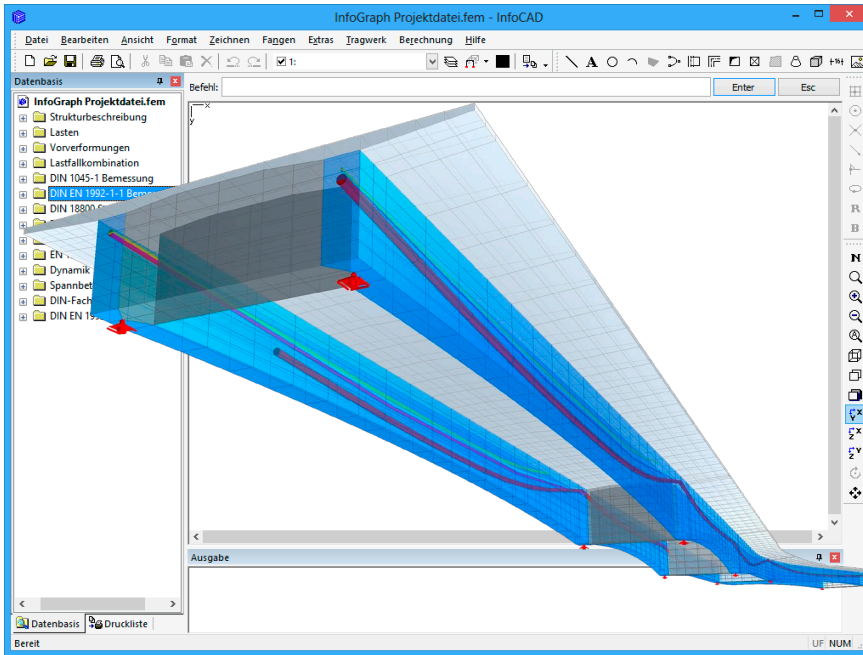
bearing safety check for the entire system (e.g., for movable frames with regard to effective rigidities or to determine deformations in condition II).



## Design Situations according to EN 1992-1-1

Situation	Ultimate Limit State	Section	Serviceability Limit State	Section
Permanent, Temporary Accidental Earthquake	Longitudinal reinforcement Lateral reinforcement Torsional reinforcement	6.1 6.2 6.3		
Characteristic (rare)	Robustness reinforcement (on the basis of EN 1992-2, 6.1(110))	9.2.1.1	Concrete compressive stresses Reinforcing steel stresses Prestressing steel stresses Crack width, prestress with direct bound	7.2 (2) 7.2 (5) 7.2 (5) 7.3.1 DE
Frequent	Fatigue simplified	6.8.6 (2)	Decompression XD1-XS3 Crack width, prestress with bound	7.3.1 7.3.1
Quasi- Permanent			Concrete compressive stresses Prestressing steel stresses Decompression XC2-XC4 Crack width, prestress without bound, Deformation	7.2 (2) 7.2 (5) DE 7.3.1 7.3.1 7.4
Fatigue	Reinforcing steel Prestressing steel Concrete	6.8.6 (1) 6.8.4 6.8.4		

# Prestressed Concrete Construction



are available, such as longitudinal or section view for editing. This makes it possible to directly evaluate the prestressing curve with regard to straining, weakening, slippage and friction. The maximum permitted prestressing force is determined using the  $\kappa$  allowance value. The forces arising from the tendon group geometry and the prestressing force curve are then applied to the system. Afterwards the calculated internal forces, deformation and other factors are available for additional checks.

## Creep and Shrinkage

The program determines concrete creep and shrinkage based on a time-dependent stress-strain law developed by Trost.

## Program Concept

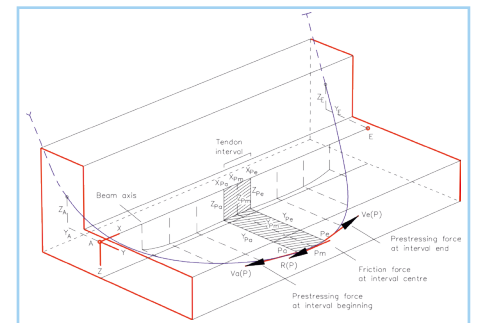
The Prestressed Concrete module was developed for all applications of prestressing with subsequent bond or without bond, which means it is particularly well suited for calculating prestressed concrete bridges, containers and floor slabs. The module is based on a 3D tendon guide that can be used for all beam, shell and solid models. The functions are divided into tendon group input, load processing in the FEM analysis section and checks

according to EN 1992-1-1, EN 1992-2, DIN 1045-1, DIN Technical Report, OENORM and SIA.

## Actions from Prestressing

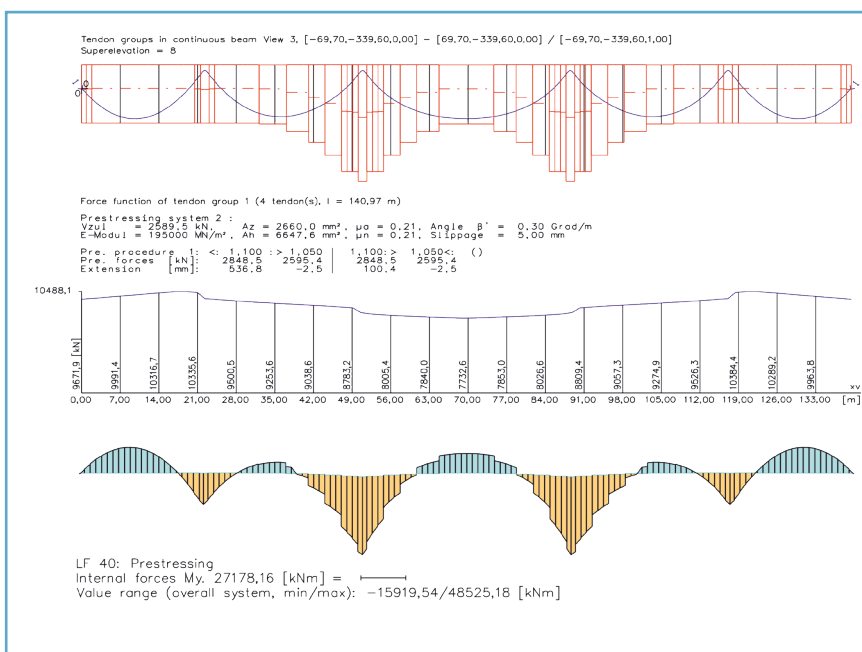
Tendon groups are entered interactively on the existing structure model. These groups are defined independent of elements, which allows them to take any course through the structure and be freely copied or moved.

For designing the tendon group geometry, a variety of help options



Load introduction at the beam element

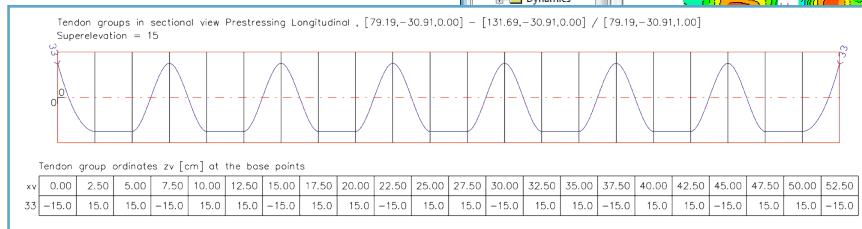
The prestressing steel layers are included in the calculation of the creep and shrinkage load case while the entire rigidity matrix is being processed. This results in composite elements whose strain state is taken to determine the corresponding share of internal forces of the composite components. This approach is implemented for all element types. Creep redistribution can therefore also be determined for area and solid models. The necessary creep and shrinkage coefficients can optionally be calculated.



## Checks According to EN 1992

The checks can be used for all prestressed engineering constructions.

Various components can be com-



binated within the structure model:

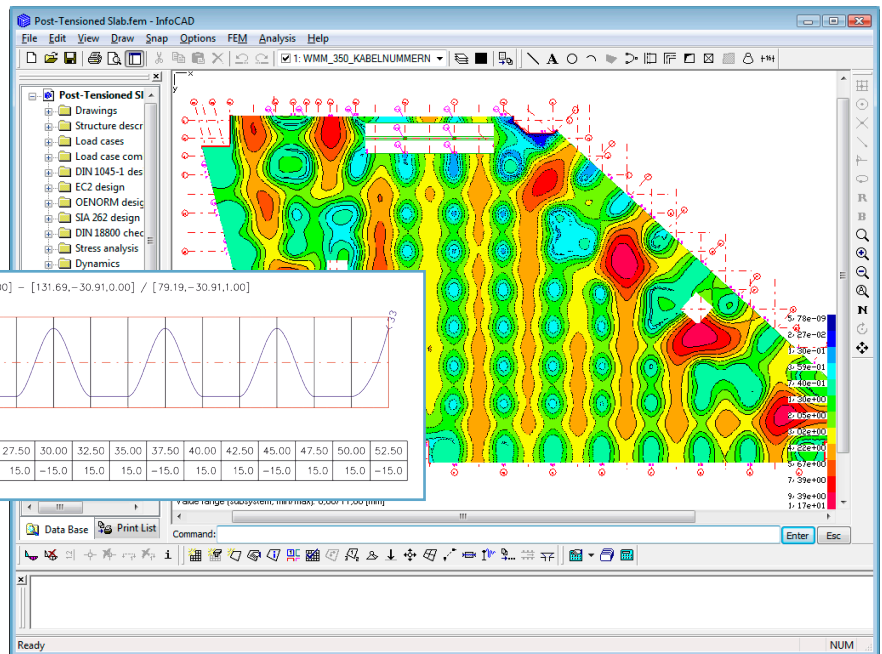
- non-prestressed components
- prestressed components with subsequent bond
- prestressed components without bond
- components with external prestressing
- mixed construction components

In addition to the checks described in the reinforced concrete section above, the fatigue of the tendons and their allowed stresses and the decompression are checked in relation to the prestressing.

These checks use additional rules for prestressed components such as the inclusion of variation coefficients for the effect from inner prestressing in the construction and final state or the bond coefficient  $\xi$ .

## Checks According to OENORM

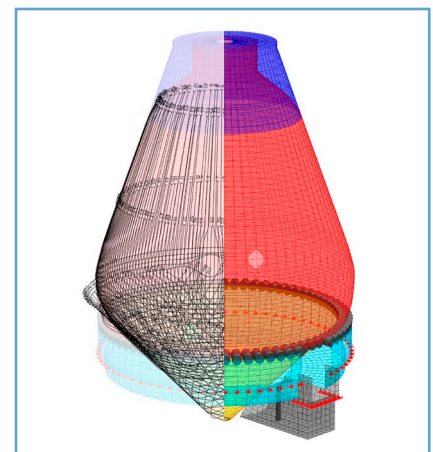
The design and checks are performed according to the individual standards OENORM EN 1992-1-1 and OENORM EN 1992-2. The regulations for general prestressed concrete construction and road



and railroad bridge construction are taken into account during these checks.

## Checks According to SIA

The regulations for general prestressed concrete construction according to SIA 262 are taken into account during the checks.



Prestressed septic tank with 3D tendon guide

## Prestressing of Beam, Shell and Solid Structures:

- Element-independent 3D tendon group guide
- Cubical spline function
- Prestressing of beam, shell and solid structures
- Tendon group editing on the system and in any section
- Friction, wobble and slippage
- Consideration of allowance value  $\kappa$
- Display of the prestressing force curve and the tendon group radii
- Identification of system reactions resulting from prestressing
- Calculation of creep and shrinkage coefficients
- Creep and shrinkage with tendon group distributions
- Design and checks according to EN 1992-1-1 and EN 1992-2 with the national annexes, DIN 1045-1, DIN-Technical Report and SIA.

# Bridge Construction



## Bridge Construction Checks

This module is used for bridge and other engineering constructions in which the actions from the road or railroad traffic have to be taken into account. Permitted structure models include 2D and 3D beam as well as area and solid constructions.

The module can be used to analyze components without prestressing as well as prestressed components with subsequent bond, without bond, with external prestressing and in mixed constructions. Optionally, the German Guideline for Recalculation of Road Bridges can be taken into account. The large number of results can be

viewed in a clear and detailed manner. A variety of graphical display options are available.

## Load Model 1

The movable traffic loads from load model 1 are processed with a special load function.

This function clearly arranges the positions of the tandem system and the UDL loads on the bridge. Any number of load positions and lane configurations can be defined in this process.

The distance between vehicles, the load distribution height and the centrifugal load of the tandem systems are automatically taken into consideration. The program

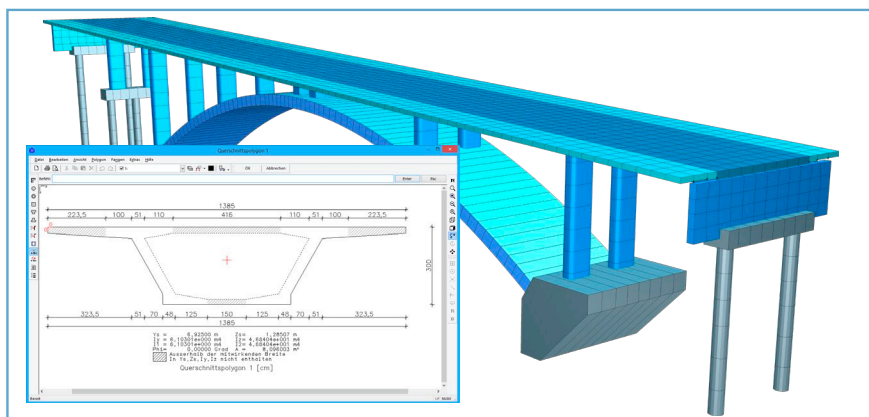
can therefore create the necessary action combinations on its own.

## Actions and Combinations

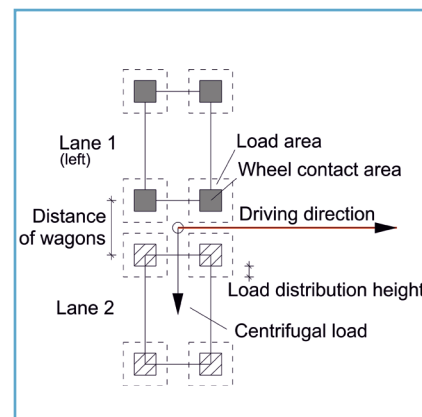
The existing load cases are assigned to the relevant actions which are then used to construct the action combinations according to the selected standard.

A variety of situations (e.g., for the construction or final state) can be created for this.

The partial safety factors and combination coefficients are specified by the program for this purpose in accordance with the applicable standard. All coefficients can be supplemented or changed during the editing process.



Model of a bridge and effective flange widths of a box girder section



Tandem system from load model 1

The program then uses this information to determine the extremal system reactions based on the combination specifications.

### Checks

Checks are carried out for each specified design situation based on the requirement classes. Depending on the situation, a variety of specifications for the construction or final state are used.

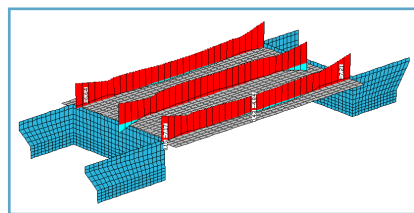
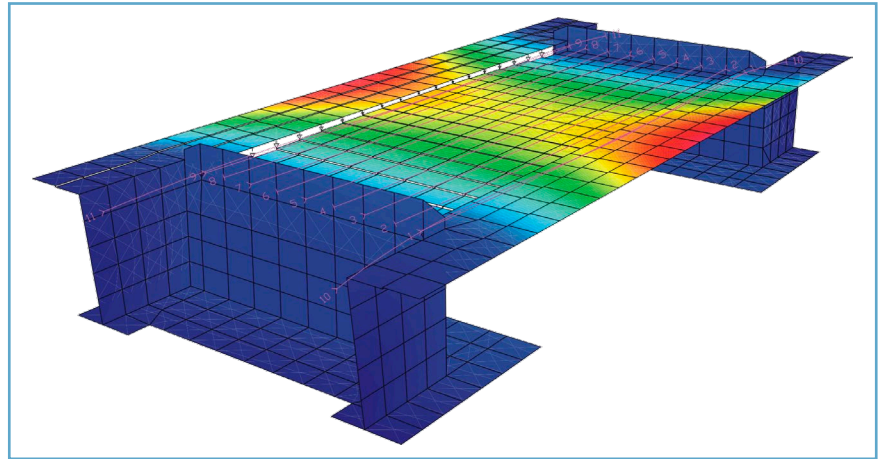
The identified reinforcements are based on the checks and may, if necessary, be increased until they are in compliance.

The longitudinal and stirrup reinforcement is stored separately based on the checks and is available for display in addition to the extreme value of all reinforcements. The checked stresses and utilizations resulting from fatigue, decompression and other factors are also stored and can then be applied to the system.

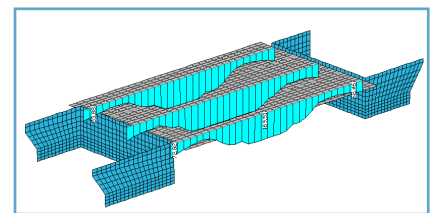
A detailed log lists all the information that is relevant for the test in a clear and easy-to-understand manner.

### Construction Stages

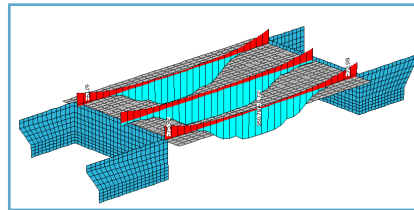
Managing construction stages is an important feature for bridge construction. It provides an easy way to determine, for example, creep and shrinkage redistributions in different phases of the construction. For every construction stage a file is created for which all calculation options are available. Redundant use of load cases and elements is prevented by the system. The results of all construction stages can be combined with or superposed on one another.



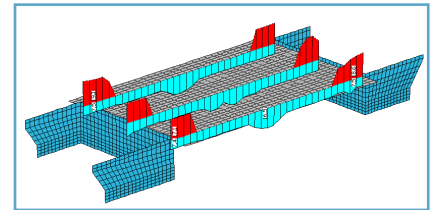
*Decompression*



*Stress range*



*Min./max. stresses*



*Extremal reinforcement*

### Bridge Construction Checks:

- EN 1992-2 and DIN Technical Report
- Consideration of any stresses and loading
- Special preparation of the load models LM 1 and LMM
- Automatic combination of actions
  - construction and final stages for all action combinations
  - optional user-defined actions
- Checks at the ultimate limit state
  - minimum reinforcement to ensure robustness
  - bending with or without longitudinal force
  - lateral force with regard to the minimum level of reinforcement
  - pure torsion and torsion with lateral force
  - fatigue (concrete, reinforcing steel and prestressing steel)
- Checks at the serviceability limit state
  - minimum reinforcement for limiting the crack width
  - limiting the crack width through direct calculation
  - decompression check
  - limiting the concrete compressive stresses
  - limiting the reinforcing steel stresses
  - limiting the prestressing steel stresses
  - check for the diagonal principal tensile stresses
- Recalculation of road bridges (Recalculation Guideline)

# Steel Construction

The steel checks according to EN 1993-1-1 can be used for polygonal beam sections made of construction steel S235 - S450 as per EN 1993-1-1, Table 3.1, as well as for cross-sections with user-defined steel properties.

For this purpose the calculated load cases are assigned to the actions in accordance with EN 1991, Part 1. Taking into account the preset safety factors and combination coefficients, defined in EN 1990, the program automatically calculates the internal forces for the desired design situations.

## Cross-Section Resistance

The elastic cross-section resistance is verified for all cross-sections of the classes 1 to 4. Cross-sections of class 4 are treated like class 3 if the c/t ratio does not exceed the limits of class 3 increased by a factor according to Chapter 5.5.2(9). Otherwise the check is carried out with effective cross-section properties as per EN 1993-1-5, clause 4.3.

The plastic cross-section resistance will be verified for all cross-sections of the classes 1 and 2 if the elastic cross-section resistance of the contemplated set of internal forces is exceeded.

$$\sigma(y, z) = N_x \frac{1}{A} + M_y \frac{\bar{z} I_z - \bar{y} I_{yz}}{I_y I_z - I_{yz}^2} + M_z \frac{\bar{y} I_y - \bar{z} I_{yz}}{I_y I_z - I_{yz}^2}$$

*Normal stresses for a section point*

## Classification

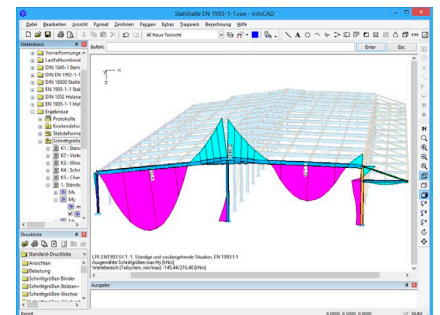
For every set of internal forces the cross-section class according to EN 1993-1-1, Chapter 5.5, is automatically determined. To this end, the stress distribution for simultaneous stress from biaxial bending and normal force in the center line of the section parts is used. A cross-section is generally classified by the most unfavorable class of its pressure-loaded section parts.

## Section Analysis

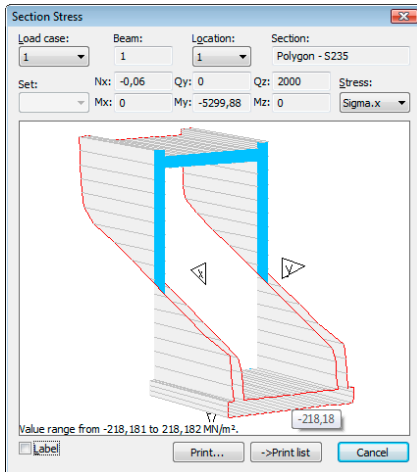
Within the scope of system checks, the following key values are determined for all used sections and then made available for use:

- centroid coordinates
- area of the section
- moments of inertia in relation to the coordinate axes and the main axes
- deviation moment
- twisting angle of the main axes
- torsion moment of inertia
- shear values for  $Q_y$ ,  $Q_z$  and  $M_x$  and the  $W_y$ ,  $W_{zy}$  and  $W_{qz}$  moments of resistance.

To determine the stressing as a result of St. Venant torsion, the differential equation of the warping







Stress curve at a polygon section

function is solved, and to establish the stressing as a result of lateral force, the differential equation of the shear warping is solved. The analysis is carried out for any thick or thin-walled polygon section and all library sections.

### Stability Analyses

Two basic methods are available for analyzing stability problems:

- Calculating structures based on the second-order theory.
- Determining buckling eigenmodes with the relevant branching load factors.

For system calculation based on the second-order theory (equilibrium on the deformed system), a check is performed for beam buckling and toppling as well as for area element buckling. The calculation can take into account additional nonlinear effects such as the exclusion of tensile stresses for bedding and support, the failure of tensile or compression beams and nonlinear material behavior.

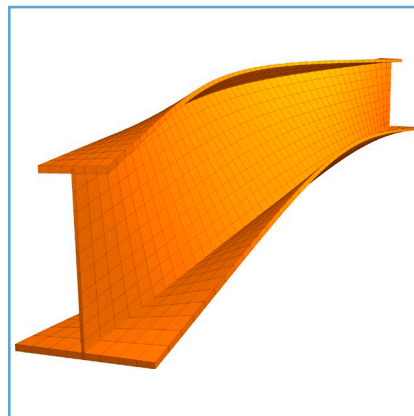
The iteration method normally converges after a few steps. Stability failures are displayed by the singularity of the entire rigidity matrix. Determining the buckling eigenmodes allows you to define the

branching load factors directly. The construction can then easily be evaluated using the associated failure figure.

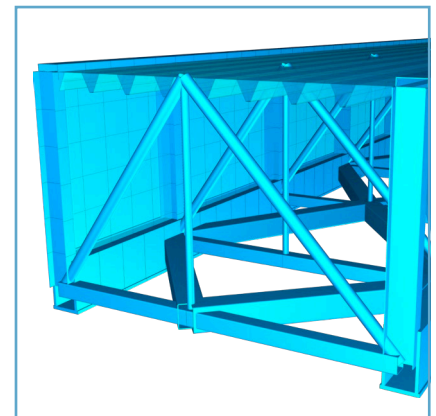
### Lateral Torsional Buckling

The lateral torsional buckling problem is solved for single- and double-symmetric I and U profiles. To create the required strain, specify the internal forces of the beam ends and a point load or uniformly distributed load in both principal directions. The ideal lateral torsional buckling moment  $M_{cr}$  is calcula-

ted by varying the elastic potential. The resulting eigenvalue problem provides the smallest positive load factor and thus the sought-after lateral torsional buckling moment. This has the advantage that the user does not need to specify the moment coefficient  $\zeta$ . Load eccentricity and elastic rotational bedding can be defined by the user. The calculation yields a detailed log of the results with system diagrams and all formulas required for the verification.



Lateral Torsional Buckling of a FEM-Model



Visualization of a dam bridge

### Options for Steel Construction:

- First-order and second-order theory, cable structures based on third-order theory
- Buckling eigenmodes with branching load factor
- Consideration of predeformation
- Analysis of single beam failure
- Superpositions and checks according to EN 1993-1-1
- Elastic-elastic, elastic-plastic and plastic-plastic
- Automatic classification and design of section classes 1-4
- Section analysis for any polygon (thick- or thin-walled)
- Spring joints
- Eccentric beam connections
- Compression and tension beams
- Extensive profile library
- Custom-defined user database
- Lateral torsional buckling check
- DXF, DSTV and IFC import/export

# Dynamics



Sophisticated constructions require both static and dynamic structure analyses. These analyses can range from resonant frequency determination and time-step calculations to nonlinear cable dynamics. The Dynamics program module lets you analyze 2D and 3D beam, cable, area and solid models.

## **Resonant Frequency Determination**

Determining the eigenvalues and eigenvectors forms the basis for most dynamic analyses. The distribution of mass that these analyses take into account is the result of the system geometry, the additional point masses and the equivalent masses of the selected load cases.

## **Earthquake Checks**

In this check, the soil acceleration and the calculation coefficients are specified according to the selected standard for any given beam or shell structure depending on the earthquake zone, the subsoil, the damping and the structure class.

The check then calculates the structure reactions associated with the resonant frequencies based on the response spectrum method and superposes them according to the SRSS or CQC method. The resulting internal forces are then available for further superposition with the static load cases and subsequent design. In order to evaluate the modes of vibration to be considered, the applied masses are contrasted with the effective modal masses. If necessary, you can define custom response spectrums.

## **Pushover Analysis**

The pushover analysis is a non-linear method for detection of the seismic safety of buildings. The calculation presents the capacity curve in the multi-mass system and in spectral representation.

## **Vibration Calculations**

Time-step calculations are carried out for periodic and transient load-time curves and specified node accelerations. These calculations

can factor in exciters of variable frequency at the same time. You can select the number and duration of the individual time steps. Mass and stiffness-proportional damping, the Lehr's damping measure, Rayleigh's system damping and/or single viscous dampers can be used depending on the calculation. At selected nodes the system response can be calculated using a fixed frequency range in the steady state (stationary response).

## **Dynamic Train Crossing**

Dynamic train crossing is another interesting calculation option provided by the program. This option offers an easy way to analyze the dynamic stress for any beam and shell structure based on predefined trains such as the ICE or Thalys or user-defined trains.

The description of the tracks is carried out by entering any continuous line on the structure. Multiple tracks can be considered simultaneously, so that, for example, the effect of trains traveling in opposite

directions have on each other can be investigated.

The load model of the typical trains for high-speed lines are stored in the program. Additionally, user-defined train loads can be added. Further default parameters are the speed and the departure time for each train. Every train is assigned to one track. By varying the departure times, the intervals between the different trains can be defined.

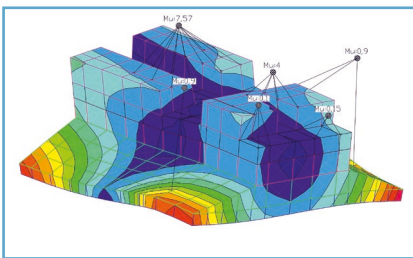
The dynamic train crossing can be analyzed as part of a direct or modal time step integration. The amount and duration of the time steps can be entered by the user.

The vibration behavior and the desired route of the train are considered here.

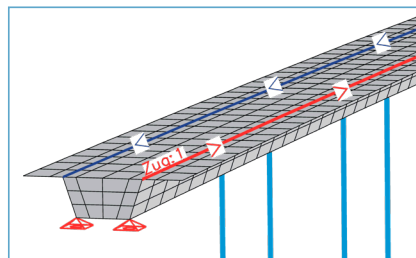
### Results Display and Processing

Deformations, speeds, accelerations, internal forces and support reactions for each time step are provided as calculation results. These can be represented individually or as time-step diagrams. Additionally, the deformation can be dynamically animated in the System Viewer.

For further analyses, the results can be superposed with reactions from static calculations and then designed.



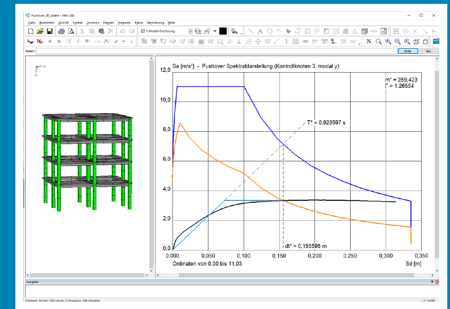
Eigenmode of a machine foundation



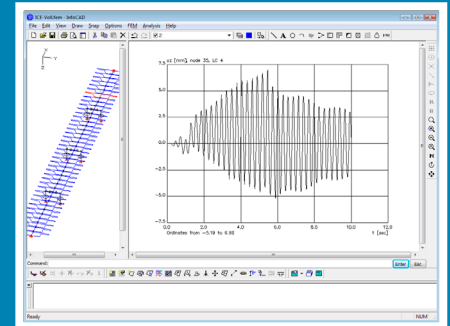
Lane of a train crossing

### Dynamic Analyses:

- Calculation of beam, cable, area and solid models
- Determination of eigenvalues and eigenvectors
- Distributed dead loads, point masses and masses from loads
- Periodic and free load-time curves
- Simultaneous variable exciters at any node
- Free node accelerations
- Lehr's damping, mass and stiffness-proportional damping, Rayleigh's system damping and single viscous dampers
- Modal analyses and direct integration of motion equations
- Time-step integration for all system reactions with selectable iteration steps
- Stationary response
- Evaluation of response spectra acc. to EC 8, DIN, OENORM and SIA
- ZPA method for the consideration of missing effective masses
- User-defined response spectra
- Calculation and output of effective modal masses
- Dynamic train crossing on any track
- Collapse analysis e.g. component failure
- Animation of all time-step calculations



Result of a pushover analysis



Displacement  $u_z$  of a selected node

Settings

EC2 OENORM SIA 262 DIN FB DIN 18800  
FEM Dynamics Load Case Combination DIN 1045-1

Analysis mode:  
Response spectrum DIN 4149:2005

Eigenvalues  
Number of eigenvalues: 10  
Max. vector space: 18 Iteration accuracy: 1 [%]

Soil accelerations  
ax: 0.8 [m/s<sup>2</sup>]  Raise components separately  
ay: 0.8 [m/s<sup>2</sup>] Reference sys. angle [°] 0  
az: 0 [m/s<sup>2</sup>]

Coefficients of DIN 4149:2005  
Subsoil circumstances: A-R Lehr's damping measure ksi: 0.05  
Subsoil parameter S: 1 Importance coefficient gamma I: 1  
Response spectrum control periods TB: 0.05 TC: 0.2 TD: 2 Behavior coefficient (horiz.) q: 1.5  
Combination method: Standard

Store static pseudo loads

OK Cancel Apply Help

Input dialog for earthquake checks

Train crossing

Label: ICE 3 Direction Colog No.: 1

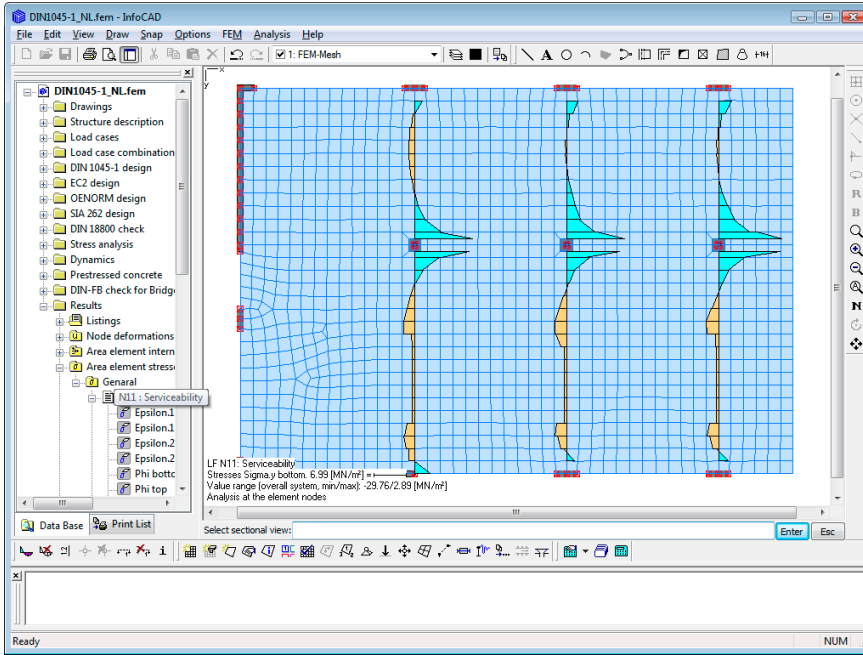
Train: ICE 3 v: [km/h] 325 Start time: 0

	[m]	Axle load [kN]
ETR-Y-500		
Eurostar		
FAALS 151		
FAL		
HSLM-A01	0	149.4
HSLM-A02	2.5	149.4
HSLM-A03	7.375	148.4
HSLM-A04	9.875	148.4
HSLM-A05	4.775	166.3
HSLM-A06	7.275	166.3
HSLM-A07	42.15	163.3
HSLM-A08		
HSLM-A09		
HSLM-A10		
IC		
ICE 1		
ICE 2		
ICE 3		

OK Cancel

Train definition with speed specification and starting time

# Nonlinear System Analysis



Concrete tensile strength (tension stiffening) that remains after the initial crack formation in a slab.

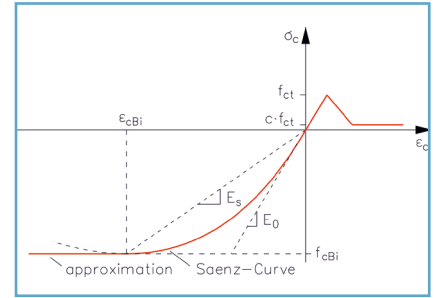
Nonlinear system analysis can be used to determine the internal forces and deformation values of 2D and 3D beam and shell structures made of reinforced concrete and steel under consideration of geometric and physical nonlinearities. For solid elements, the stress state is calculated according to the Raghava, Rankine, Drucker-Prager or Lubliner yield condition. In addition, the damage models according to Mazars, De Vree and an elastoplastic damage model (Lubliner, Lee & Fenves) are available. The program is especially well-suited for checking the ultimate limit state (buckling safety check) and the serviceability (deformations, internal forces) of the entire system.

## Structures Made of Reinforced Concrete

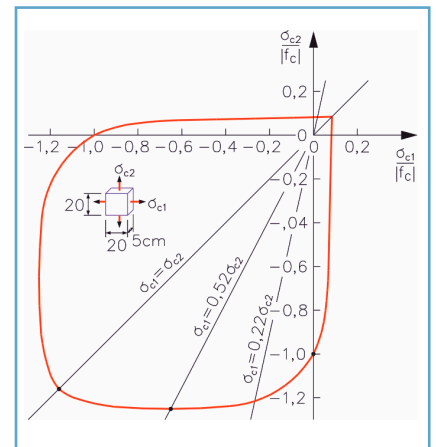
The stress-strain curves for nonlinear internal force calculation are applied based on the corresponding standard and the material selected. The corresponding material partial safety factors are also taken into

consideration. The concrete tensile strength can be considered with softening or bilinear behavior. This calculation is based on the reinforcing steel from a previous design. Alternatively, you can also specify the reinforcement level directly.

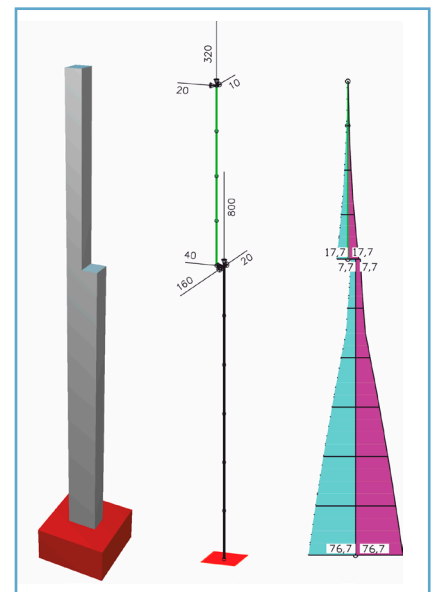
The consideration of concrete creep is realized by modifying the underlying stress-strain curves. For frameworks, the reinforcement can be automatically increased so as to arrive at the desired structural safety or to comply with the permitted strain. The biaxial concrete behavior is implemented for area elements under consideration of the strengths according to Kupfer/Hilsdorf/Rüsch and the concept of equivalent one-axis strain. In addition to the deformations and internal forces in state II, the remaining concrete and reinforcing steel stresses and its strains are output.



Standardized equivalent stress-strain curve for concrete under multiple-axes strain



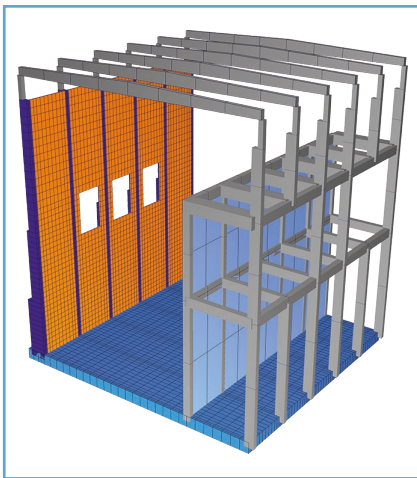
Biaxial failure curve according to Kupfer/Hilsdorf/Rüsch



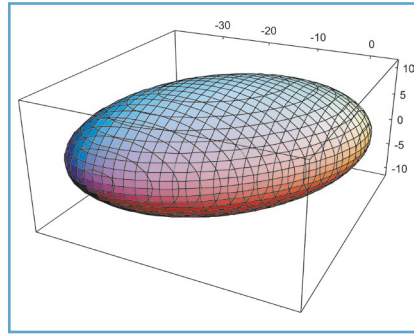
Reinforcement distribution according to load-bearing safety check

## Structures Made of Steel and Bilinear Material

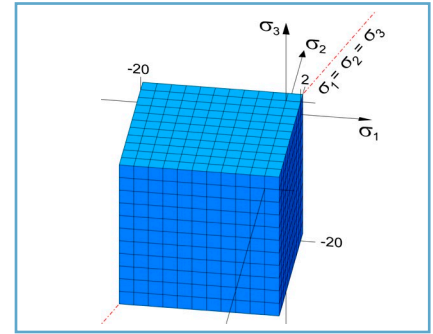
The calculation is carried out according to the theory of plasticity. The check guarantees that, under consideration of the internal force interaction, the full-plastic internal force limits are not exceeded and that the system is in a stable state of equilibrium. In addition to the internal forces and deformations, the nonlinear stress and strain distribution in the structure is made available.



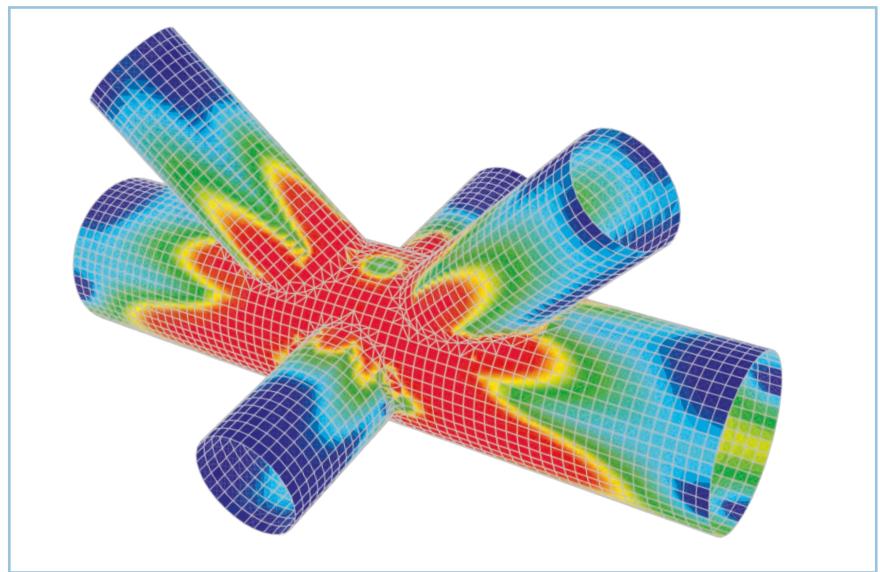
Reduction of the constrain forces by the cracking at a waste bunker



Yield criterion according to Raghava



Yield criterion according to Rankine



The plasticized areas can be seen clearly on the depicted tube joint. The loading capacity has almost been reached with the existing strain.

### Nonlinear System Analysis:

- Reinforced concrete with stress-strain-curve according to EN 1992, DIN, OENORM and SIA 262
- Effect of concrete on tension between cracks
- Ultimate limit state
  - Check under consideration of existing reinforcement
- Serviceability limit state
  - Deformations under consideration of existing reinforcement
- Consideration of long-time deformations as result of concrete creeping
- Steel with bilinear stress-strain curve under consideration of the Huber-von Mises yield criterion and interaction with all internal forces
- Bilinear stress-strain curve and individually definable compressive and tensile strength (Raghava, Rankine, Drucker-Prager or Lubliner yield criterion)
- Damage models according Mazars and De Vree and elasto-plastic damage (Lubliner, Lee & Fenves)
- Soil Model according to Mohr-Coulomb
- Calculations in the supercritical range using the arc length method
- Calculation of load-displacement curves
- Equilibrium on the deformed system according to second-order theory and advanced geometrical nonlinear theory
- Bedding with bilinear bedding curve for frameworks
- Automatic reinforcement increase during ultimate limit state check for frameworks

# Structural Analysis for Fire Scenarios

With this module you can conduct a Structural Analysis for Fire Scenarios based on the general calculation method for 2D and 3D beam and shell structures.

Steel, reinforced concrete, timber and composite sections can be considered.

The following calculations are available for solid models:

- stationary and instationary temperature distributions
- user-defined heat sources
- stress analysis considering the strains due to a thermal action

## Thermal Calculation

The temperature distributions in the sections are determined as part of a nonlinear time-step integration. Initially the requisite heat transfer conditions and fire stresses are assigned to the section edges. The unit temperature-time curve, user-defined fire curves or a constant ambient temperature can be selected in this process.

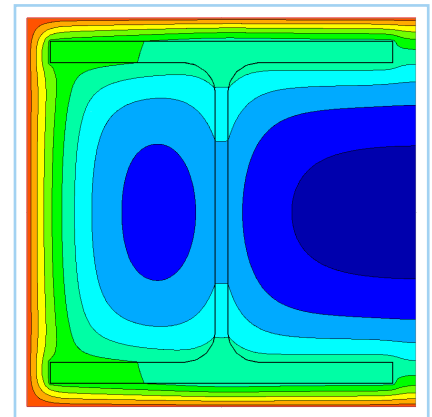
The result is a temperature profile for all specified times.

The specified time increment and duration are used for the thermal analysis and the subsequent mechanical analysis.

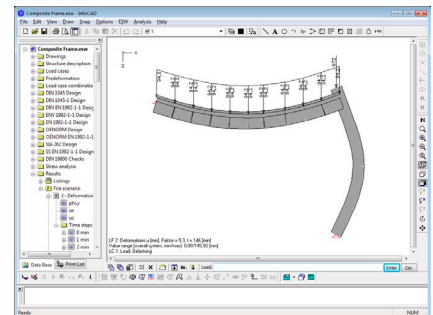
## Mechanical Analysis

As part of the mechanical analysis, a nonlinear time-step calculation is performed based on the 'General Calculation Method' defined in Chapter 4.3 of the EN 1992-1-2.

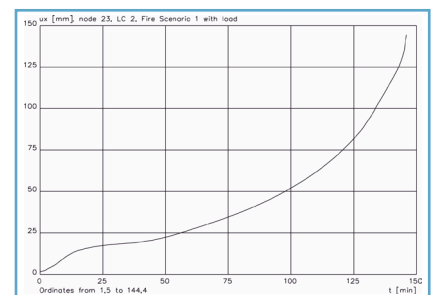
The thermal strains and stress-strain curves in the section are determined using the temperature profiles from the thermal analysis. Afterwards the time-dependent deformations, the internal forces and the support reactions are available.



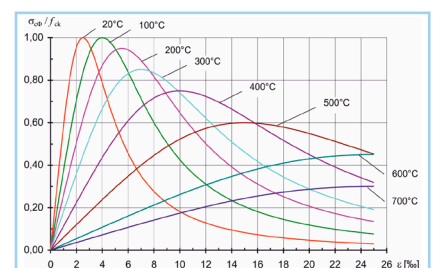
Composite section with flame applied to three sides



Deformation at a specified time step



Time-displacement curve at a specified system node



Related stress-strain curves for concrete with quartz additives

## Structural analysis for fire scenarios:

- Any kind of section geometry
- Steel, reinforced concrete and free materials (e.g., insulation) within a section
- Predefined and user-defined thermal material properties
- Unit temperature-time curve, hydrocarbon fire curve etc.
- User-defined fire curves, different fire scenarios
- Calculation of the temperature profile in the section via nonlinear time-step integration
- Heat transfer by radiation and convection in cavities
- Temperature-dependent thermal strains and stress-strain curves
- Nonlinear time-step calculation of the structure based on the general calculation method
- Graphical display of the time-dependent deformations, internal forces and support reactions
- Thermal calculation of solid models

# Timber Construction

The timber checks are designed for buildings and engineering constructions under observance of the EN 1995-1-1, EN 1995-1-2 and DIN 1052 for beam elements.

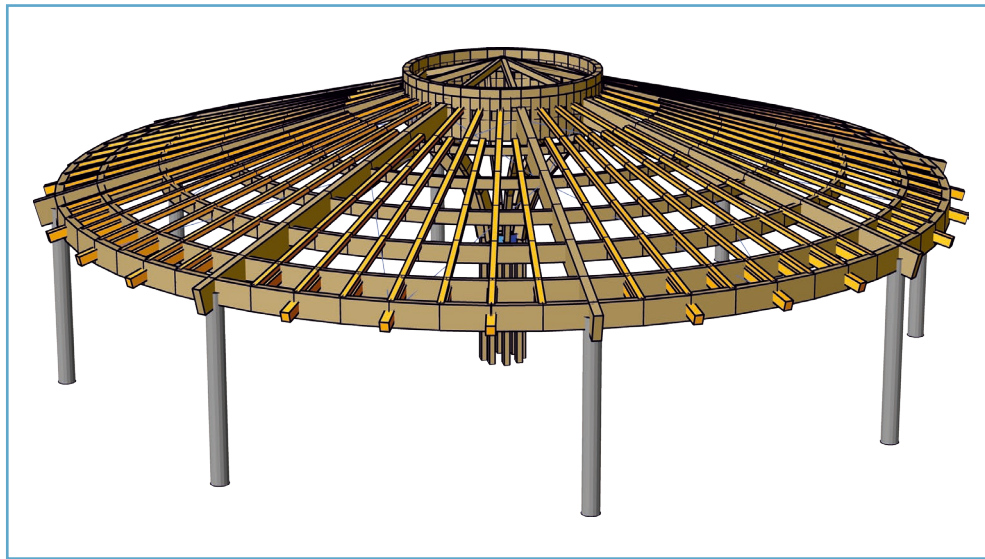
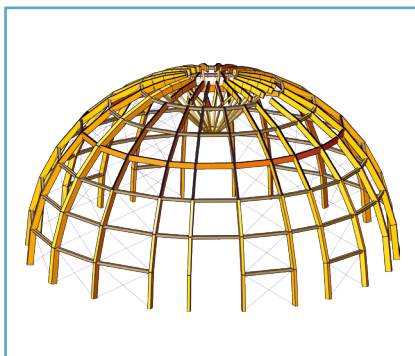
## Actions, Design Situations

The checks are carried out after the static calculation. For this purpose the load cases are assigned to the actions. Taking into account the preset safety factors and combination coefficients defined in EN 1990, DIN 1052 or DIN 1055-100, the program automatically calculates the internal forces for the desired design situations.

## Cross-Section Checks

For every set of internal forces the modification factor  $k_{mod}$  will be determined from the service class and the load-duration. Then the following checks will be performed:

- Tension parallel to the grain
- Compression parallel to the grain
- Bending
- Combined bending and axial tension
- Combined bending and axial compression
- Shear from lateral force
- Torsion
- Combined shear and torsion



## Buckling Check

The buckling checks are carried out according to the equivalent beam method as described in DIN 1052, Chapter 10.3 and EN 1995, Chapter 6.3.2, respectively.

## Checks for the Deflections of Beams

In accordance with Chapter 7.2, the deflections of beams should not exceed the values in Table 7.2. For this purpose, the instantaneous deflection is determined for the characteristic and the final deflection for the quasi-permanent design situation.

## Design Method for Fire Conditions

Structural fire design is carried out according to the reduced cross-section method as described in EN 1995-1-2, Chapter 4.2.2. For this,

the flame-edges, the duration of the fire and the charring rate  $\beta_n$  are evaluated.

## Utilization, Results

The utilization is defined as the ratio between the action  $E_d$  and the resistance  $R_d$  of a cross-section.

The following results are available:

- Utilization of the beams for each situation.
- Maximal utilization of the beams of all situations.
- Maximal utilization of the sections of all situations.

Additionally the extremal design values for internal forces, support reactions, deformations, soil pressures and stresses are saved for all check situations. The detailed log also lists the decisive combination internal forces of all design situations for each result location.

## Timber Checks:

- EN 1995-1-1 and EN 1995-1-2, DIN 1052
- Automatic combination
- Check of every set of internal forces considering the modification factor  $k_{mod}$
- Checks under fire conditions
- Integrated buckling check
- Checks for the deflections of beams

# Solid Models



Solid elements are used if a structure geometry, which is to be analyzed, cannot be described adequately by beam or area elements or if the ascertainment of the three-dimensional stresses is necessary for the evaluation of the structural behavior. Examples are the dynamic analysis of machine foundations or the calculation of concrete bridges with a complex geometry. The ability to model realistic support and connection conditions with contacts or nonlinear bedding also makes solid elements useful.

## Modeling

Model objects are used to define a structure model. They consist of freely combinable partial solids that are created by the extrusion of sections or by entering polyhedrons with four to eight corners. Object properties like material, color, layer etc. can be assigned immediately. Bedding and contact properties can be defined at each model surface. Afterwards the complete solid model is meshed with tetrahedron elements taking into account all boundary conditions.

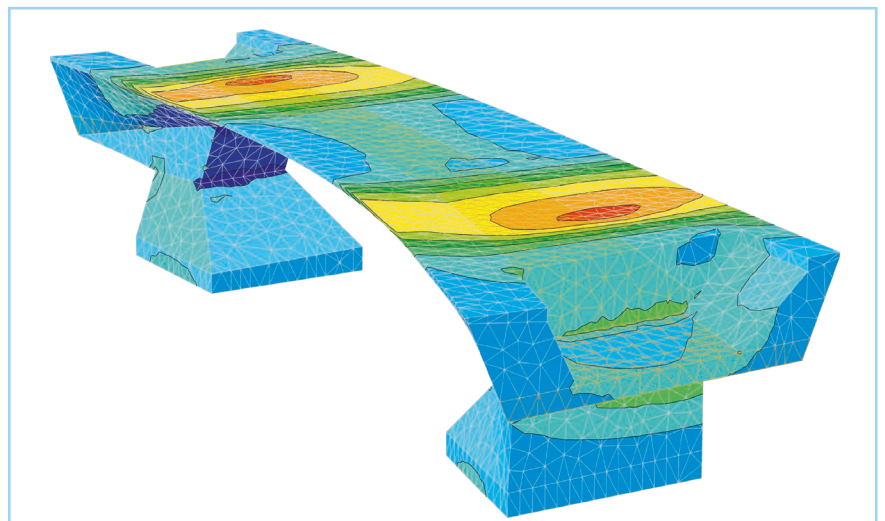
## Loads

All necessary load types are available for the analysis of solid models:

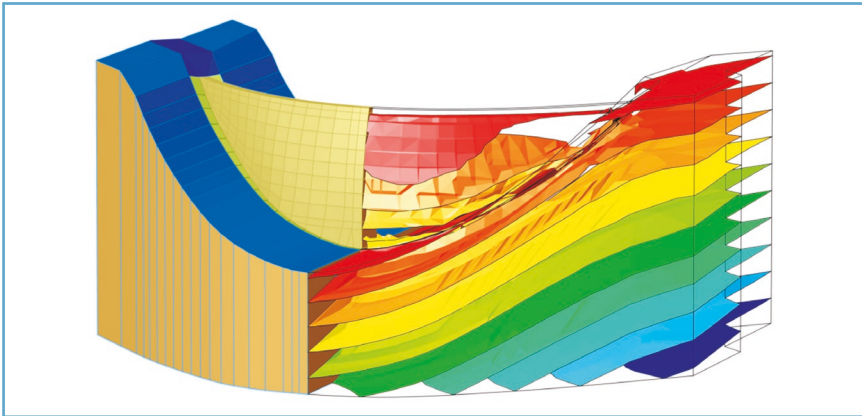
- Dead load and nodal loads
- Free point, line and area loads
- Support displacement
- Linear temperature fields
- Free tendon layout / prestressing
- Creep and shrinkage
- Load model 1 for bridge constructions
- Dynamic train load

## Analysis

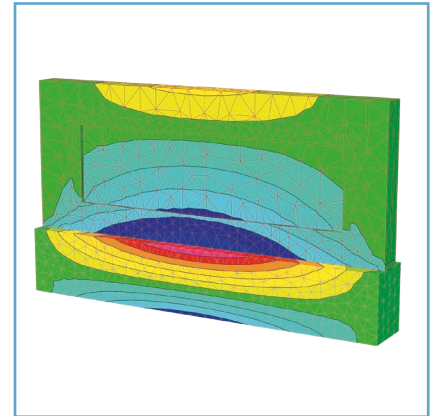
For the FEM analysis a tetrahedron element with 10 nodes is available, which can exactly describe linear stress distributions and hence has a very good convergence of results. Also a high computation speed is associated with this element. Additionally to the static and dynamic abilities mentioned above, stability analyses (second order theory, buckling, bulging etc.), contact elements and plastic material behavior (Huber-von Mises, Raghava, Rankine, Drucker-Prager) as well as damage models are implemented.







*Dam wall with rock foundation – isosurfaces of stress distribution*



*Stress distribution as a result of difference in temperature*

### Thermal Calculation

This allows the determination of transient temperature distributions in solid models.

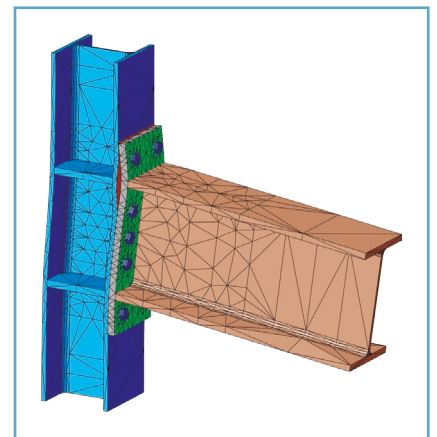
The thermal properties are assigned to the solid faces. For the determined temperature distributions, the stresses and the strains can be calculated.

### Results

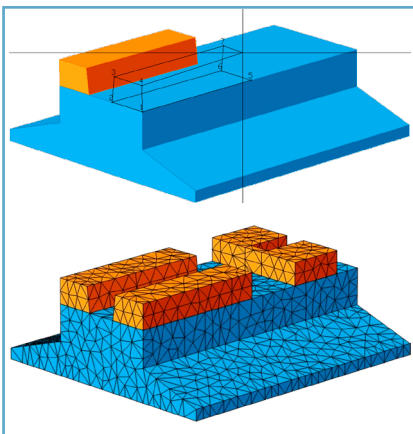
The possibilities of result preparation are important to evaluate the quality of a calculation. They are necessary to understand and document the structural behavior. Among others the following result representations are available for solid models:

- Deformation with animation
- Color surfaces
- Three-dimensional isosurfaces
- Surface sections
- Solid sections
- Principal stress vectors
- Integral internal forces for checks
- Temperature distribution

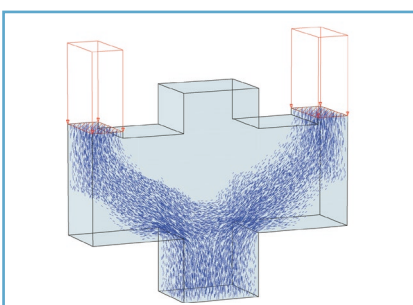
After integration of the stresses with design objects, internal forces are available for the checks.



*Head plate connection with contact conditions*



*Solid model and tetrahedron mesh*

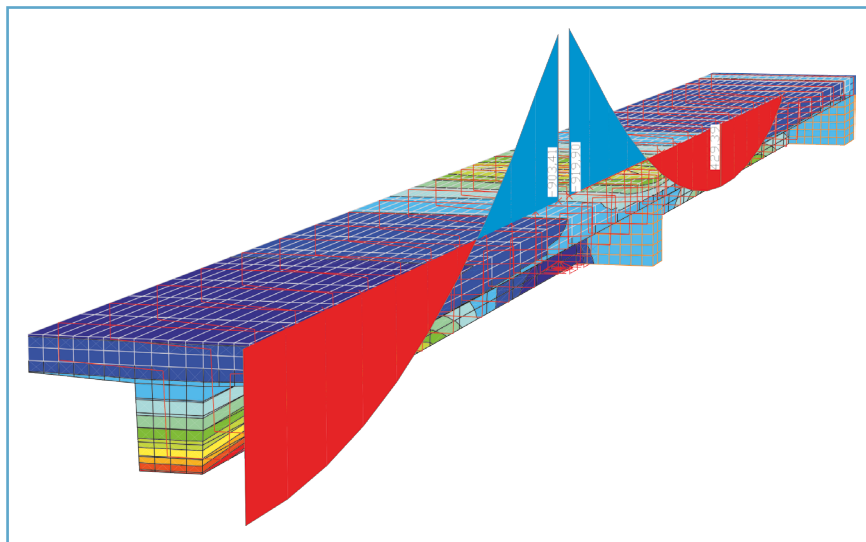


*Principal stress vectors*

### Solid Models:

- Structure generation with model objects
- Definition with extrusion of sections or polyhedron objects
- Direct mesh generation of IFC objects (BREP)
- Automatic mesh generation with local refinement
- 8 and 10 node elements
- Free point, line and area loads
- Linear temperature fields
- Free tendon layout / prestressing
- Stability Analysis
- Nonlinear supports and bedding
- Contact
- Plasticity theory (Huber-von Mises, Raghava, Rankine, Drucker-Prager, Lubliner)
- Damage according to Mazars, De Vree or elasto-plastic (Lubliner, Lee & Fenves)
- Thermal Analysis

# Design Objects



*Solid model with total internal forces*

## Design Objects

Solid construction frequently relies on calculation models that consist of a combination of beam and area elements or solid elements. These models let you view various construction stages and the resulting creep redistribution within a section.

Since the models determine internal forces separately for beam and area elements, they cannot be used directly for reinforced concrete or prestressed concrete checks. The design objects can be used to add up the reactions of the individual elements into 'total internal forces,' which are in turn provided to the checking programs for additional processing.

## Integration

The reactions are integrated for all elements located in a section of the design object. The integrated internal forces refer to the axis of the design object.

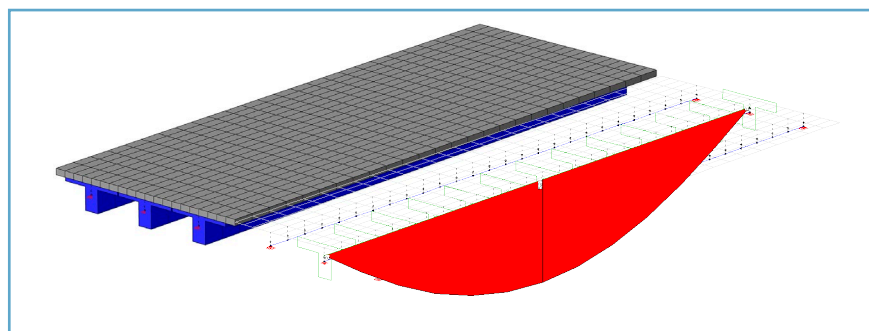
They are then made available for each load case for display and processing purposes.

## Solid Structure Checks

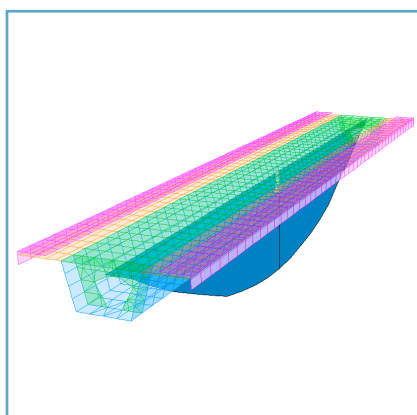
All reinforced concrete and prestressed concrete checks can be carried out for the sections of the design objects. All of the design specifications are preset depending on which section is being checked. Thanks to this innovation, it is now also possible to design prestressed solid structures.

## Composite Sections

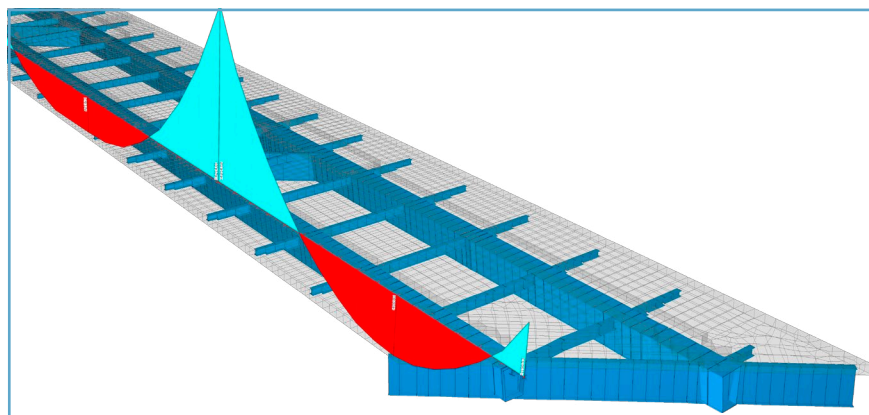
To handle sections with different materials, composite sections can be assigned to the design objects. These sections consist of multiple subsections. A single design can be performed for composite sections made of steel and concrete.



*Pre-cast girder with cast-in-place slab and integrated forces*

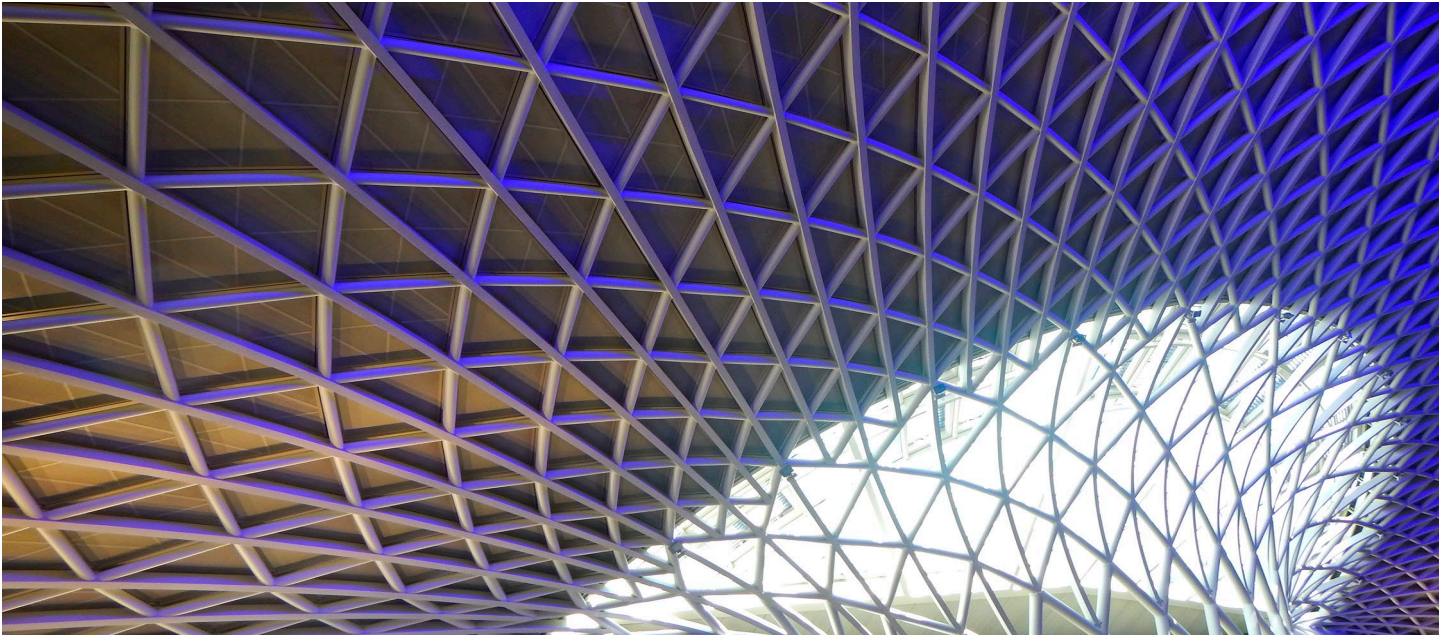
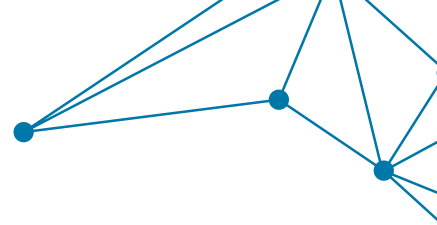


*Integration of an area model*



*Total internal forces of slab and main girders*

# NURBS

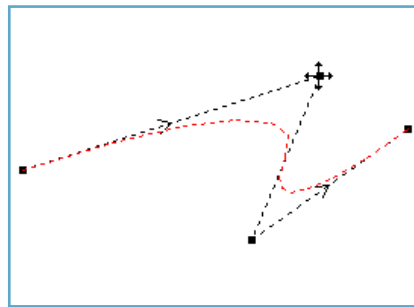


## Freeform Structures Using NURBS

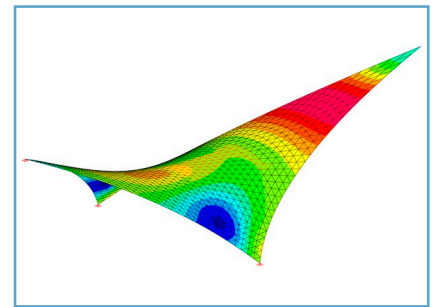
You can enter freeform structures using NURBS curves, areas and solids (**N**on-**U**niform **R**ational **B**asic **S**pline). This makes it easy to create structures that do not conform with any particular shape defined by ruled geometry. Also, element meshes can be instantly generated on the NURBS.

A NURBS curve is described by a control polygon. The level of the NURBS indicates how closely it approaches the polygon. By simply moving the control points, the NURBS can be transformed into practically any shape.

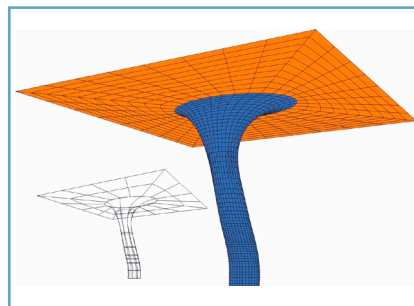
NURBS areas are normally created using multiple profile curves or edge curves. Moreover, you can generate any rotational surface or extrude solids. Numerous functions make the handling of highly complex structures simple.



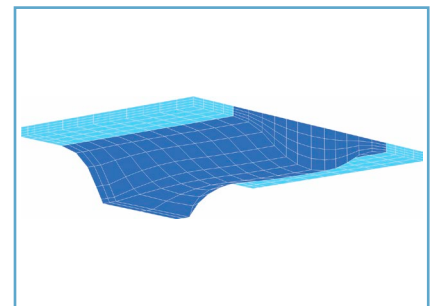
*NURBS curve with control points*



*FEM mesh with stress distribution*



*NURBS area and FEM mesh*



*Solid model*

## Freeform Geometries with NURBS:

- NURBS curves, areas and solids
- Specification of control and interpolation points
- Generation of NURBS areas from profile curves
- NURBS areas from four edge curves (Coons patch)
- Insertion of additional nodes in individual directions
- Connection of NURBS
- NURBS created through rotation of curves or areas
- Automatic mesh generation based on area and solid elements

# Data and Prices

## Features of the InfoCAD Basic Modules

### System Processing

---

Interactive 3D CAD user interface with layer functions  
CAD functions such as copy, mirror, move and others for all objects  
NURBS objects for handling free-form geometries  
Drawing objects as construction aids  
Automatic and manual dimensioning  
Fully automatic mesh generation for 2D and 3D systems with an option for manual partial meshes  
Intersections and generation of ruled surfaces  
Manual element editing including condensation, adjustment, node movement, etc.  
No restriction on directions of internal forces and reinforcements  
Independent mesh controls  
Automatic alignment of element eccentricities  
Line joints, beam joints with spring rigidities, link joints and single links  
Spring elements with any nonlinear characteristic (FEM)  
Elastic bedding with exclusion of tensile stresses with a bedding factor or the modulus of compressibility method  
Rigid or spring-mounted single and line supports, exclusion of tensile supports (FEM)

### Calculation

---

Predefined concrete qualities, steel, timber, user-definable materials  
Polygonal sections, steel construction profile library, custom profile database  
Tension and compression beams  
Management of construction stages  
Element-independent point, line and area loads, support displacement, temperature, hydrostatic pressure (FEM)  
Generation of moving loads, determination of influence lines and surfaces  
Internal force and stress determination  
Design objects for stress integration at any section  
Load case combination with min/max determination of all system reactions  
Actions and situations for ultimate and serviceability limit states  
Combination information for defining the load cases involved in a result with their respective weighting

### Solid Structure Checks

---

Minimum reinforcement to ensure robustness, bending with/without longitudinal force/longitudinal force only  
Lateral force, torsion and torsion with lateral force, checks against punching shear  
Check of the shear joint, fatigue for concrete and reinforcing steel, crack check

### Steel Construction Checks

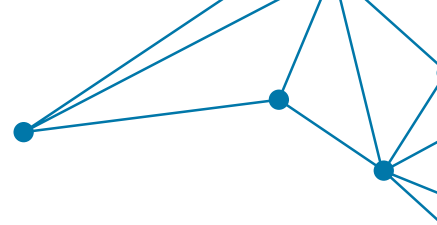
---

Stresses and utilization for section classes 1-4 acc. to EN 1993-1-1  
General section analysis for polygonal beam sections

### Result Output and Data Exchange

---

Graphical and tabular views, descriptive visualization and animation  
Serial print with automatic updating of results plots  
DXF interface, Windows clipboard, DSTV and IFC data transfer (BIM), BCF editor  
BIM add-in for Autodesk® Revit®, transfer of reinforcements to construction programs



## Price Overview of the InfoCAD Basic Modules and Add-Ons

### Basic Modules

---

<b>Finite Elements 2D</b> .....	<b>3,900.00 €</b>
Slabs and plain stress elements with downstand beams, 2D frameworks based on the second-order theory and girder grids.	
<b>Finite Elements 3D</b> .....	<b>8,950.00 €</b>
Slabs, plain stress elements, shells and prismatic shell structures as well as 2D and 3D frameworks based on the second-order theory.	
<b>2D Frames</b> .....	<b>1,900.00 €</b>
<b>3D Frames</b> .....	<b>2,500.00 €</b>
<b>Axisymmetric Shells</b> .....	<b>2,300.00 €</b>
Rotationally symmetric shell structures based on the finite element method.	

### Add-Ons

---

<b>Serviceability Checks Reinforced Concrete</b> .....	<b>750,00 €</b>	
Limiting concrete compressive stresses, limiting reinforcing steel stresses, minimum reinforcement for the crack width limitation, limiting the crack width through direct calculation.		
<b>Timber Checks</b> .....	<b>750,00 €</b>	
<b>Cable Elements</b> .....	<b>1.500,00 €</b>	
Cable structures with or without prestressing according to the theory of large deformations.		
<b>Second-Order Theory</b> .....	<b>1,250.00 €</b>	
Calculation of shell structures and solid models based on the second-order theory, determination of buckling eigenmodes incl. load factors.		
<b>Equation Solver</b> .....	<b>1,250.00 €</b>	
Parallel Sparse Solver, substructure technique and iterative equation solver for large systems.		
<b>Lateral Torsional Buckling Check</b> .....	<b>500.00 €</b>	
Check based on the equivalent beam method for single- and double-symmetrical I and U profiles.		
<b>Solid Structures / Models</b> .....	<b>2,950.00 €</b>	
<b>Dynamics</b> .....	<b>3,900.00 €</b>	
<b>Prestressing</b> .....	<b>4,900.00 €</b>	
<b>Bridge Construction Checks</b> .....	<b>3,900.00 €</b>	
Checks for beam, area and solid structures (Special price in conjunction with the 'Prestressing' module) .....		<b>2,900.00 €</b>
<b>Nonlinear System Analysis</b>		
Extension for the '2D Frame' program module .....		<b>1,500.00 €</b>
Extension for the '3D Frame' program module.....		<b>2,000.00 €</b>
Extension for the 'Finite Elements 3D' program module.....		<b>3,500.00 €</b>
<b>Structural Analysis for Fire Scenarios</b>		
Extension for the 'Nonlinear System Analysis' program module .....		<b>2,450.00 €</b>
<b>InfoCAD BIM RV</b>		
BIM add-in for Autodesk® Revit® .....		<b>free</b>



## General

### Scope of Delivery

---

The InfoCAD program system is delivered together with the user manual on CD. It is compatible with Windows 8 and newer. The programs feature software protection and are installed with single-user or network licenses. The licensing fees do not depend on the type of installation.

### Pricing, Installation

---

All goods and services are subject to the General Terms and Conditions of InfoGraph GmbH. The listed prices are quoted ex works Aachen, Germany, and exclude VAT. On-site installation will be charged on a time and material basis.

### Multiple Licenses

---

The price list shows the fees for the initial license of the program components. In the case of multiple installations at one business location, the following scale of discount shall apply:

2nd license:	40% discount on initial license
3rd or subsequent license:	70% discount on initial license

### Warranty Period

---

The warranty period is 6 months. Within that period the licensee may receive program maintenance and application support free of charge.

### Software Maintenance Agreement

---

Upon conclusion of a software maintenance agreement, the licensee is eligible for all program revisions and extensions during the entire agreement period. In addition, the licensee may make use of the support services of the licensor regarding program system use at any time via telephone, mail or the Internet.

### Extension of Program Licenses

---

All program licenses can be extended in a modular fashion. Program components purchased during the warranty or maintenance period shall be credited based on the range of functions they provide. After this period, the licensee may be charged for updating older program components prior to extension.



## Consulting

Investing in software represents a long-term decision for a particular line of products. Qualified consulting is playing an increasingly important role when making such financial decisions, particularly with respect to new generations of standards. By exclusively employing civil engineers with years of professional experience, we are able to better understand your needs and concerns and provide you with the best consulting services available.

## Maintenance

The product must be able to meet the complex requirements of real-world use at all times. To meet this demand and to ensure long-term performance, we continuously improve the product and adapt it to rapidly changing conditions.

Last update: April 2024

### Photo Credits

---

Stock Fotos from 123rf.com: Image 30695480, Author: Lev Kropotov, Image 11558331, Author: Dmytro Surkov, Image 3679075, Author: Iurii Konoval, Image 7746636, Author: Emanuel Kaplinsky, Image 34179967, Author: Pan Demin, Image 39741873, Author: PaylessImages, iStockphoto.com: Image 19814856, Author: Sara Winter

**InfoGraph GmbH**

Kackertstraße 10  
D-52072 Aachen, Germany

Tel.: +49 241 88 99 80  
Fax: +49 241 88 99 88 8

[info@infograph.de](mailto:info@infograph.de)  
[www.infograph.de](http://www.infograph.de)

