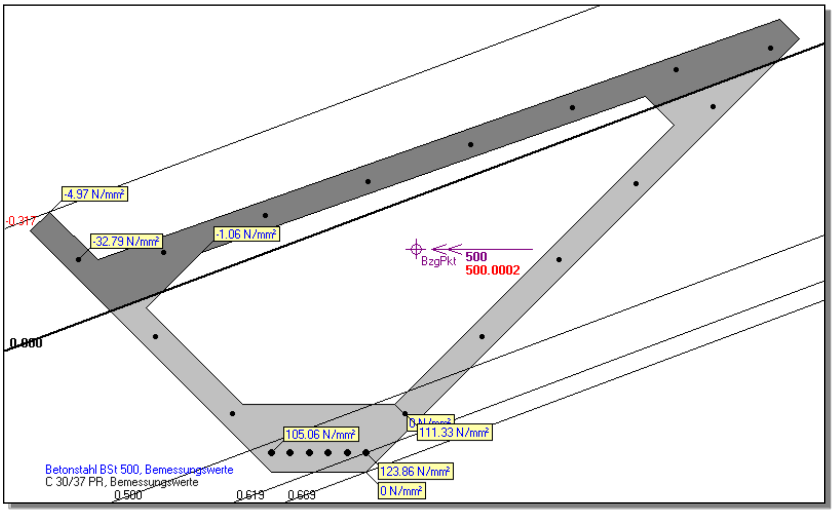
INCA2



Program description

INCA2, Version 3.0

Last updated: March 2024

Author: Dr.-Ing., Uwe Pfeiffer
Am Schulland 15, 21224 Rosengarten
pfeiffer@tuhh.de
www.u-pfeiffer.de

Table of contents

1 Introduction

This program is used for the interactive calculation of arbitrarily composed cross-sections under biaxial bending with longitudinal force; especially reinforced concrete cross-sections. The calculation is based on linear or nonlinear stress-strain-relationships. Prestressing, shrinkage and creep, the stiffening effect of the concrete in the cracked tensile zone or subsequently supplemented cross-sections can be taken into account.

Due to the general formulation of the calculation algorithms, it is possible not only to calculate cross-sections of concrete and reinforcing steel, but also to use any other material such as wood or plastics, as long as the material properties can be modelled with the stress-strain-relationship. To choose the correct material properties, please read the help point Stress-Strain-Relationship / Material Properties.

The cross-sections can be modelled as a polygon of straight-lined edged surfaces (e.g. concrete) or large recesses in such surfaces, and secondly of points with a diameter (e.g. reinforcement) or small recesses (e.g. cladding pipes).

The external forces are either related to the concrete center of gravity, the ideal center of gravity or any other point.

Detailed explanations of the basic calculation method (stress integration, iteration method) are given by Busjaeger, Quast (1990): Program-controlled calculation of arbitrary concrete cross-sections under two-axis bending with longitudinal force (program MasQue), issue 415 of the German Committee for Reinforced Concrete, Beuth Verlag GmbH, Berlin-Cologne.

2 Input/Output System and Load Case

2.1 Cross-sectional file

All input values for the INCA2 program are stored in a file with the extension ***.inc** . If the data of the current file is changed by the user, a security prompt for saving is displayed when a new file is opened or when the program is closed.

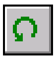
From left to right:

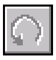
New	an empty file is created; pre-defined materials of the file ***Standard-Baustoffe.inc*** are loaded, Shortcut: Ctrl + N
Open	a dialog to open an INCA2 file appears, Shortcut: Ctrl + O
Save	a dialog to save an INCA2 file appears, Shortcut: Ctrl + S

In addition, the ***Save As*** item is available in the ***File menu***.

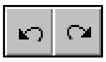
The data in the ***.inc** file is stored in binary format. In order to be able to read the data (such as coordinates of points, polygons, load cases) as text, please select the menu item ***File*** with the sub-item ***Input Data (num.)***.

Especially for demonstration purposes (lectures, etc.), it makes sense to be able to quickly restore a cross-sectional file to its original state after changes have been made. To do this, click on the ***ReLoad button***:

 ReLoad available

 ReLoad not available (no changes after last save or file was not saved)

For working with INCA2, an ***undo*** and a ***redo*** function are also available (***undo*** and ***redo***, up to 64 times):



Undo shortcut: Ctrl + Z

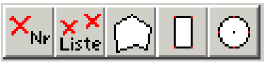
2.2 General Procedure for Modelling of a Cross-Section

The modelling of a cross-section is usually done in the following order:

1. Definition of material properties (or use of predefined materials)
2. Creating Points for Polygons
3. Connecting the points to form a polygon, assigning a material for this polygon
4. Creating points for reinforcement, assigning a diameter and a material
5. Definition of a load case N_x / M_y / M_z
6. Carrying out the calculation (strain distribution, safety factor, etc.)

Several wizards are available for creating the polygons as well as for creating standard cross-sections (rectangle, circle, I-profile, spun concrete columns), which significantly simplify the input.

The following functions are available to generate a new cross section:



From left to right:

- new point
- new points in a table / list
- creating a polygon
- creating a rectangle
- creating a circle

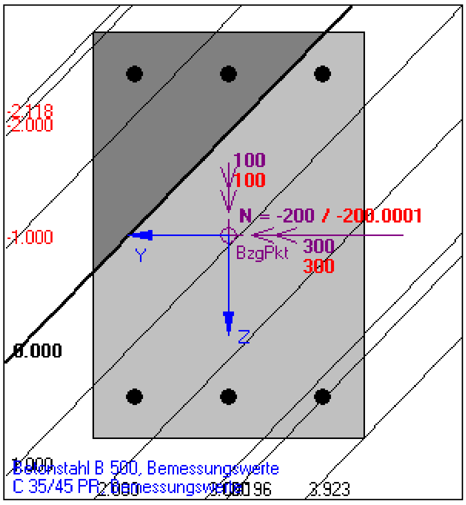
In the respective windows, one has to select the material for the concrete and for the reinforcement. One can enter a title in the ***System-Info*** sub-item. There you will also receive information about your cross-section, such as the number of points or the materials which are used.

When creating a section, you need to make sure that your cross-section contains at least one rebar (or polygon) with a “tensile-resistant” material, otherwise a calculation will not be possible. See the chapter for “Stress Strain Relationship” of this help-file for further explanations.

Coordinate system

The coordinate system is defined as follows:

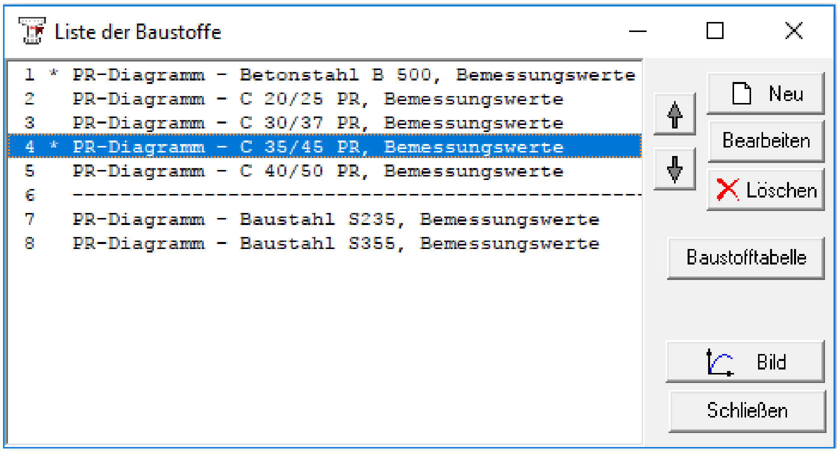
- Y-axis to the left, z-axis downwards
- Rotation angle is mathematically positively defined (counterclockwise)
- Compressive forces N_x are negative, tensile forces N_x are positive
- Positive bending moment M_y rotates around the y-axis and generates pressure on the top, tension on the bottom
- Positive bending moment M_z rotates around the z-axis and generates pressure on the left side, tension on the right side



2.3 Material Properties

2.3.1 List of materials

Menu ***Input***, sub-item ***Materials***, a list of still defined materials opens:



Within this window, materials can be created, changed or deleted. The list includes all previous materials. An asterisk at the beginning means that this material is used, followed by the type of material definition, then the name.

The buttons on the right have the following functions:

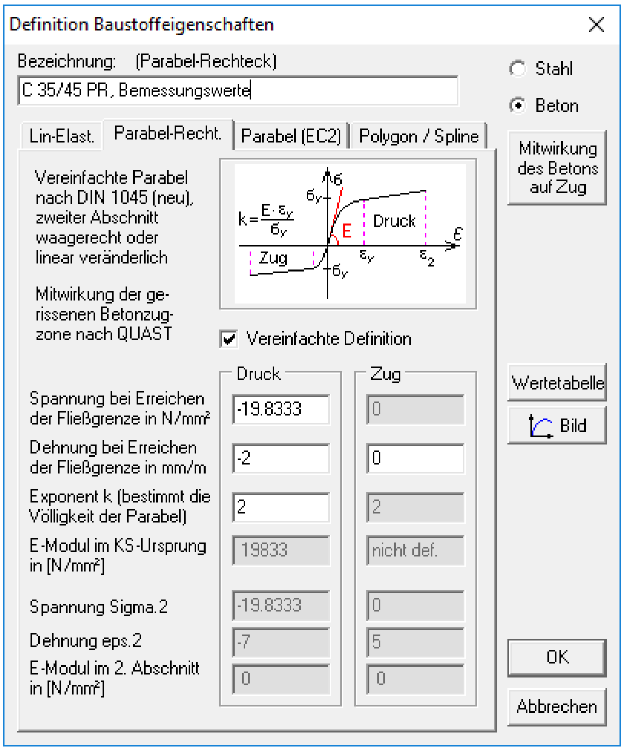
- The buttons with the ***arrows*** are used to sort the materials. Click on a material that can now be moved up or down in the list.
- New button – A new window will open in which a new material can be entered. For more information on the individual material laws, please refer to Stress Strain Lines.
- Edit button – The material marked in the list can be edited in a new window.
- Delete button – All selected materials will be deleted. If the materials to be deleted are currently being used in cross-section, a safety query is carried out before deleting. The materials can be deleted, but before calculating the cross-section, a new material must be assigned to the polygon or reinforcement.
- Button ***Table of material*** – In a new window, predefined materials are shown, which are stored in the file ***Baustoffe.inc*** . There, materials can be selected and transferred to the current INCA2 file. This file ***Baustoffe.inc*** can be opened and easily modified by the user in INCA2.
- ***Image*** button – Displays the stress-strain diagram of the selected materials in a new window. The range of values for the representation (min and max ε) results from the selected limit strains for the cross-section.
- Close ***button*** – Closes the window with the list of materials.

2.3.2 Material Laws (stress-strain curves)

For the definition of a material, the following stress-strain curves (material laws) are available, which are selected via the index card tab:

1. Linear-elastic
2. Parabola Rectangle
3. Parabola-like function according to EC2 for deformation calculations
4. Polygon / Spline

Example of a concrete C40/50, design values, with the parabolic rectangle diagram:

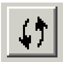


Explanations of the various material laws, the respective areas of application, advantages and disadvantages are given in chapter ***6 Modelling of materials***. Experience has shown that INCA2 has the highest number of application errors in the correct choice of materials, so users are asked to read this chapter carefully.

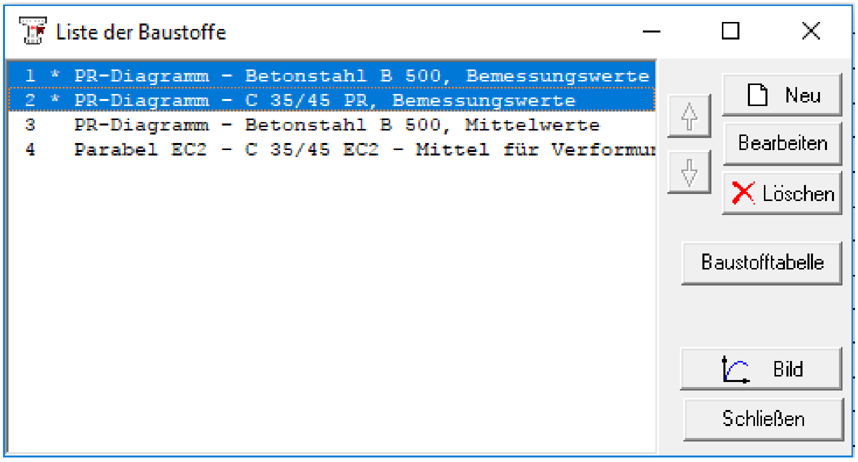
2.3.3 Switching between design values and average values

Especially in the case of non-linear calculations to determine the deformation or the internal forces, it is important to use the mean respectively the average values of material properties for the deformation calculation, but the design values for the determination of the cross-sectional load capacity. Depending on the desired type of calculation, the properties for the concrete and for the reinforcement must be changed in each case.

In order to carry out the above calculations, it might be helpful to implement two materials for the concrete and two materials for the reinforcement in the INCA2 file. In order to make the change between the materials as easy as possible, all you need to do is click on the button



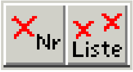
in the upper right corner of the main window of INCA2. To do this, however, the materials in the table must be sorted into groups and arranged as shown in the following window.



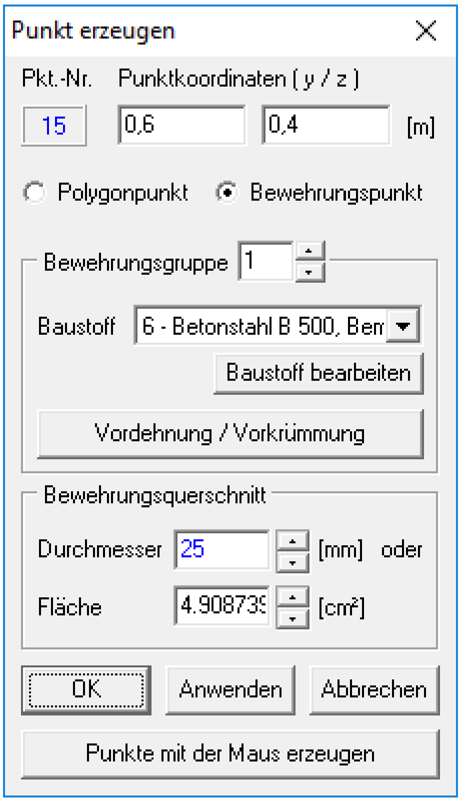
As can be seen, the design values of the material properties are arranged in the first and second positions, the mean values in the third and fourth positions. The asterisk in front of the name indicates whether the material is currently in use. By clicking on the "change button" in the main window of INCA2, the property of the concrete cross-section is changed from the design values to the mean values (No. 2 becomes No. 4), the same applies to the reinforcement (No. 1 becomes No. 3).

2.4 Input of points for polygons or as reinforcement

Via the menu ***Input***, Sub-item ***Point***, or



2.4.1 Option 1: Individual points



- Node number - is automatically specified
- Enter node coordinates y and z
- Selection of whether it is a polygon point or a rebar

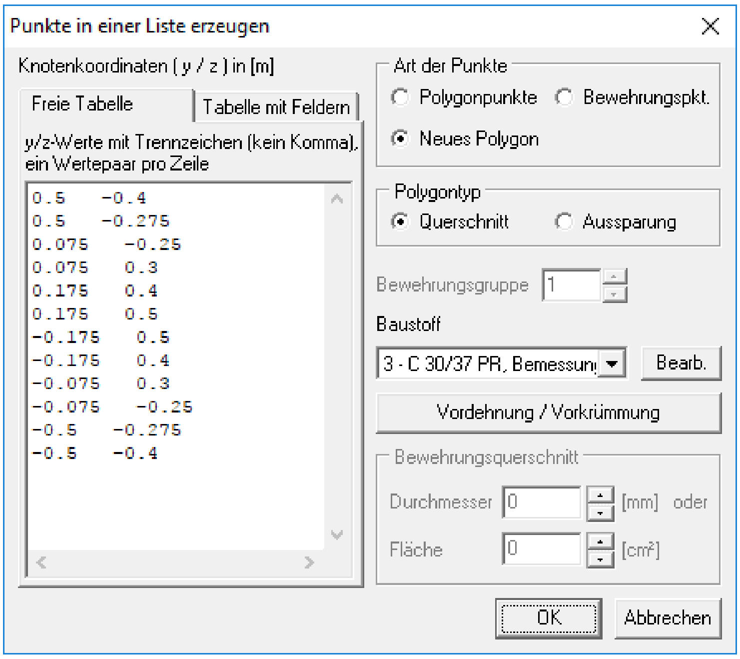
If reinforcement:

- Selection of reinforcement group:
  The reinforcement is combined into groups, each group has a material property and, if applicable, a pre-stressing. Using reinforcement groups, handling of large numbers of reinforcement is easier, because all members of a reinforcement group can be selected together and change of material can be done easier.
- Choice of a material
- Enter pre-stress / pre-curvature in an extra window (e.g. pre-stressing of steel)
- Diameter (or cross-sectional area)
- OK – node is created and window is closed
- Apply – node is created, window remains open for entering the next node
- Create points with the mouse
  - input mask closes,
  - create nodes with the left mouse button
  - return to the input mask with the right mouse button

For **Pre-stressing of steel** please read in chapter 8.2 the modelling instructions for "pre-stressing in the clamping bed" and "post-tensioning with subsequent bonding". Depending on the type of pre-stress, a slightly different pre-elongation must be defined for the prestressing steel.

2.4.2 Option 2: Points as a list

Here you can create multiple points at the same time by entering them as a free table (as shown in the following image) or in fields. It also gives you the option to immediately combine these points into a polygon.



When entering the reinforcement, you can enter either the diameter or the cross-sectional area. To make handling easier, small up-down-buttons are available in each case. However, if you want to enter negative cross-sectional areas for a recess (e.g. cladding pipe for tendons), you must enter the cross-sectional area manually.

By selecting the reinforcement group, you combine the individual members into a group, to which a material and, if necessary, a pre-stress/pre-curvature are assigned. Preferably, for example, tensile reinforcement, compressive reinforcement and structural reinforcement are combined in one group each. When designing the cross-section for a load, for example, only the tensile reinforcement can be adjusted, the structural reinforcement and the pre-stressed reinforcement remain constant.

Below a defined reinforcement point, there is usually still the concrete. In the menu under ***Tools*** => ***Settings*** => ***Results tab***, you can choose whether this concrete area should be taken into account or subtracted. In order to check a calculation result by hand, it may be useful to calculate with the gross concrete area (no recess). However, especially for higher-reinforced cross-sections with high-strength concrete, the recesses for the reinforcement should no longer be neglected and a calculation with the net concrete area should be carried out. In the case of highly reinforced columns with high-strength concretes, the load-bearing capacity can otherwise quickly be overestimated by 10% or more.

When outputting the detailed numerical results, the reinforcement points for which a recess was taken into account are marked with an asterisk (*).

If two points with the same coordinates are defined intentionally or accidentally, a warning is shown with the option not to create the point or to generate it anyway (useful e.g. in the case of cladding with internal tendons). In the graphical output on the screen, it is relatively difficult to see double respectively over-laying points. Such points can only be recognized in the numerical output.

Any number of points can be generated.

2.4.3 Modify an existing point

- Mark the point (then turns red)
- ***Edit*** menu, ***Properties*** sub-item, or
  keyboard shortcut ***Alt+Enter***, or
  double-click on the point, or Button 
- Input mask opens (coordinates, cross-sectional area)
- Make and confirm changes

Deleting an existing point

- select one or more points with the mouse
- Press the Delete key or
  Edit menu, sub-item ***Delete***

Multiple Selection - Note

If you want to change several points at the same time, you can select them together by dragging a window or by simply clicking on them with the left mouse button while holding down the Ctrl key at the same time (as in other Windows programs). If you now select the button for the properties after a multiple selection, only the fields for which the points have the same property will be filled with values.

2.5 Modelling of concrete sections (polygons)

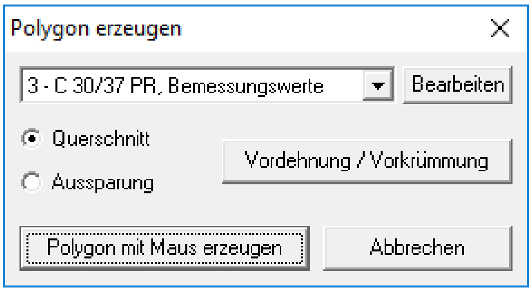
In the ***Input*** menu, you have several options for creating concrete surfaces or polygons in general:

1. Creating a Polygon
2. Create points and combine them into a polygon
3. Rectangle
4. Circle

5. Predefined cross-sections
6. Spun concrete columns

2.5.1 Polygon

With this input option, you can create an arbitrary cross-section by connecting previously defined points. Polygons must consist of at least 3 and a maximum of 100 points. In the ***Input*** menu, under ***Polygon***, the following window opens:

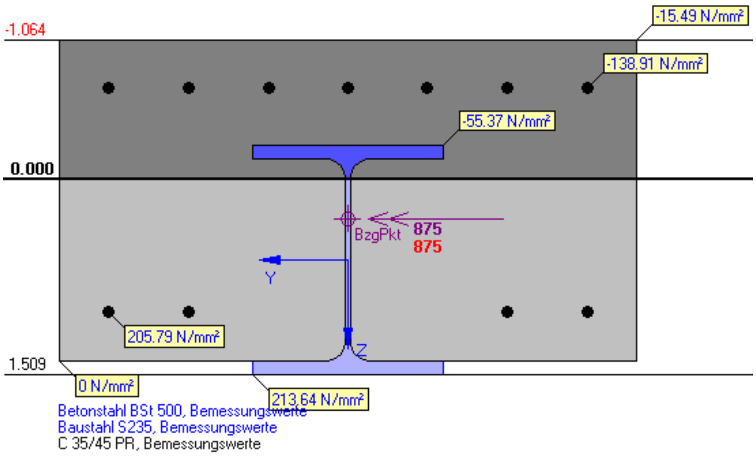


- Select material from the drop-down box (if no material has been defined yet, it must first be created)
- Select cross-section or recess
- Enter pre-elongation / pre-curvature (e.g. for subsequently added cross-sectional areas, pre-cast elements with in-situ concrete supplementation, preflex beams, etc.)
- Select Button ***Create Polygon with Mouse*** , the input mask closes
- Connect points with a mouse click, click on the first point again to close the polygon
- right mouse button to cancel the input and return to the input mask

Attention: A maximum of 100 points can be combined into a polygon! Polygons with a positive area are shown in light grey, recesses within other concrete polygons in white (without hatching). Any number of polygons can be created.

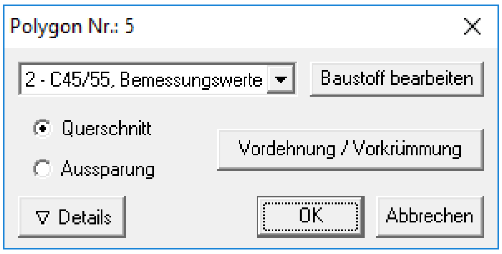
Display of polygons

For the display of polygons, a distinction is made between concrete (grey) and steel (blue). After a strain calculation, the pressure zone (in the following example on the upper side) is shown darker.

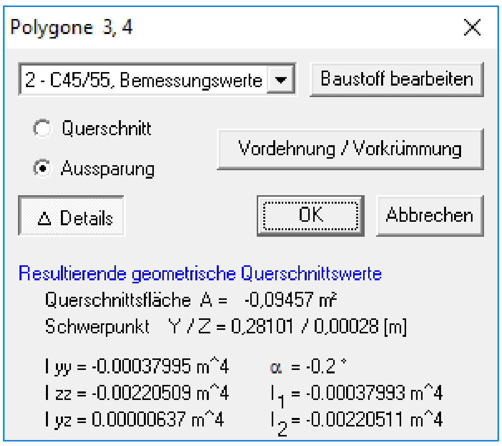


Modify an existing polygon

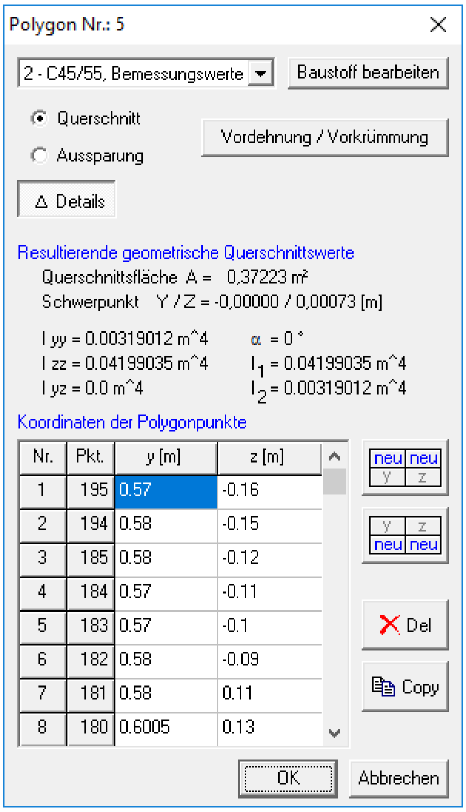
Select the polygon and select the properties button (or from the ***Edit*** menu or ***Alt+Enter***). In this window, you can change the properties of the polygon.



After clicking on the ***Details*** button, the window expands. If the properties of multiple polygons are displayed, the cross-section values of the selected polygon surfaces are displayed under Details.



If the properties of only one polygon are displayed, the polygon points can also be edited, new ones can be added, or points can be removed from the polygon.



Furthermore, you also have the option of adding points to the polygon in the graphical interface. To do this, select the desired polygon (it will then be outlined in red). Now move the mouse to the desired place where you want to insert another point in the polygon. After clicking on the right mouse button, a pull-down menu opens, in which you select the second item ***Insert point***. If you have activated the grid in the settings, the new point will be created for the next "round" grid coordinates.

Deleting an Existing Polygon

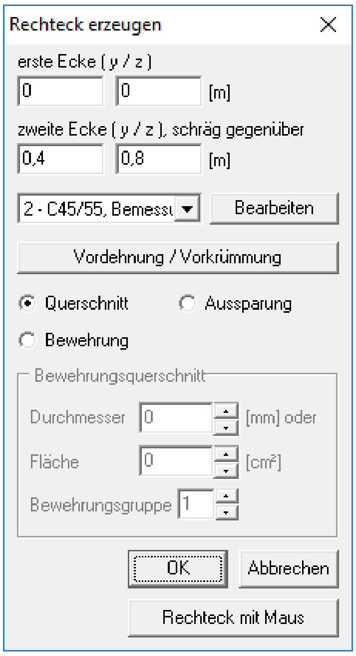
Select Polygon and press Delete (or in the ***Edit menu***, sub-item ***Delete***).

2.5.2 Polygon as a list of points

See chapter *2.4.2* Option *2: Points as a list*.

2.5.3 Polygon as Rectangle

Menu ***Input***, sub-item ***Rectangle***, the following window opens:

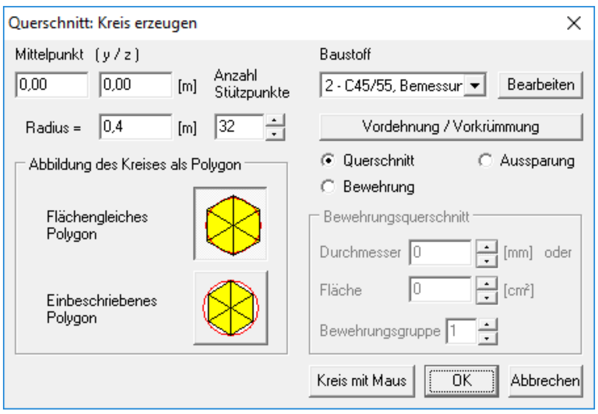


You define the rectangle by entering the coordinates of two opposite corners. Material and pre-stress / pre-curvature can be chosen. Then you choose whether it is a cross-section, a recess or 4 reinforcement points. For this latter, please also specify the diameter and group of reinforcement.

If you do not want to specify the coordinates numerically, you can use the ***Rectangle with mouse button*** to create the rectangle directly on the drawing area.

2.5.4 Polygon as Circle

Menu ***Input***, sub-item ***circle***, the following window opens:



You define a circle by specifying its centre and radius. Since the circle is mapped as a polygon, a higher number of points result in higher accuracy, but also increase the computation time. To get a fully symmetrical polygon, the number of points should always be divisible by 4.

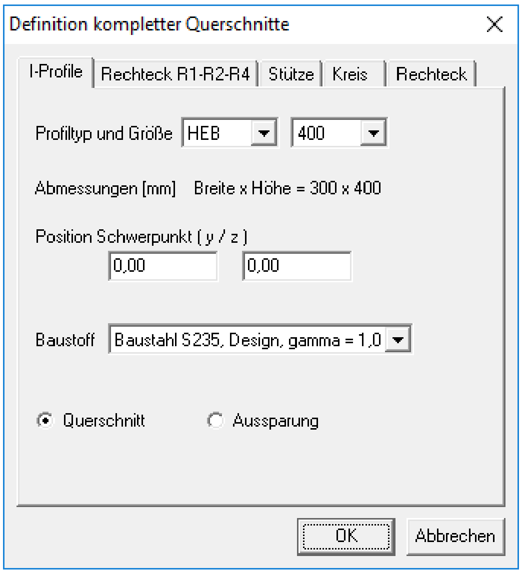
If you want to create a cross-section or a recess, it is recommended to use the option as an "***equal-area polygon***". The area of the circle with the selected diameter and the area of the polygon are exactly the same, the individual edge points are therefore slightly outside the circle.

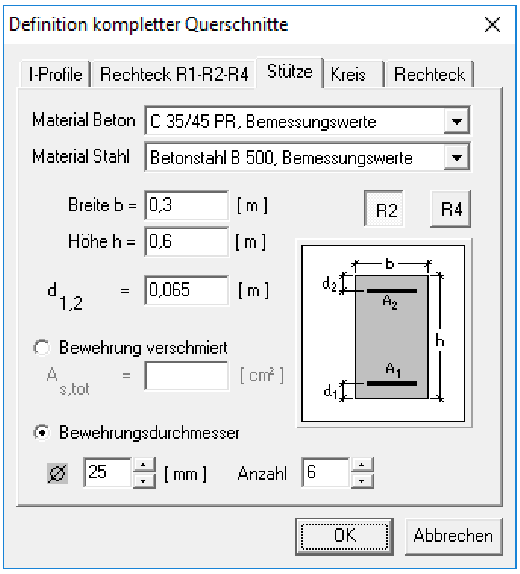
However, if you want to create individual reinforcement points, please use the option "***Inscribed polygon***", in this case the points are created exactly on the edge of the circle.

If you do not want to specify the coordinates and radius numerically, you can use the "***Circle with mouse*** " button to create the circle directly on the drawing area.

2.5.5 Wizards for complete cross sections

Menu ***Input***, sub-item ***Complete cross-sections***, the following window opens:





In this window, you can quickly and easily create various steel construction profiles up to rectangular and circular cross-sections by choosing the several tabs.

I-Profiles (steel sections)

Select the desired profile type and size, as well as the other input values. The profile data is stored in the ***Profile.txt*** file, which can be changed but not expanded.

Rectangle R1-R2-R4

Here, a rectangular cross-section is created with smeared reinforcement in each layer. The reinforcement quantities and edge distances for the individual layers can be specified separately (in this respect, the options R2 and R4 only refers to the position of the reinforcement and is therefore not entirely correct). The smeared reinforcement consists of 10 reinforcement bars per layer, which are arranged in such a way that the cross-sectional area and moment of inertia are equal to that of an equivalent rectangular cross-section.

Column (R2-R4)

If the reinforcement is arranged symmetrical (e.g. for columns), this option is preferred. In addition, it is possible to specify the number and diameter of the reinforcing bars.

Circle (circular ring / tube)

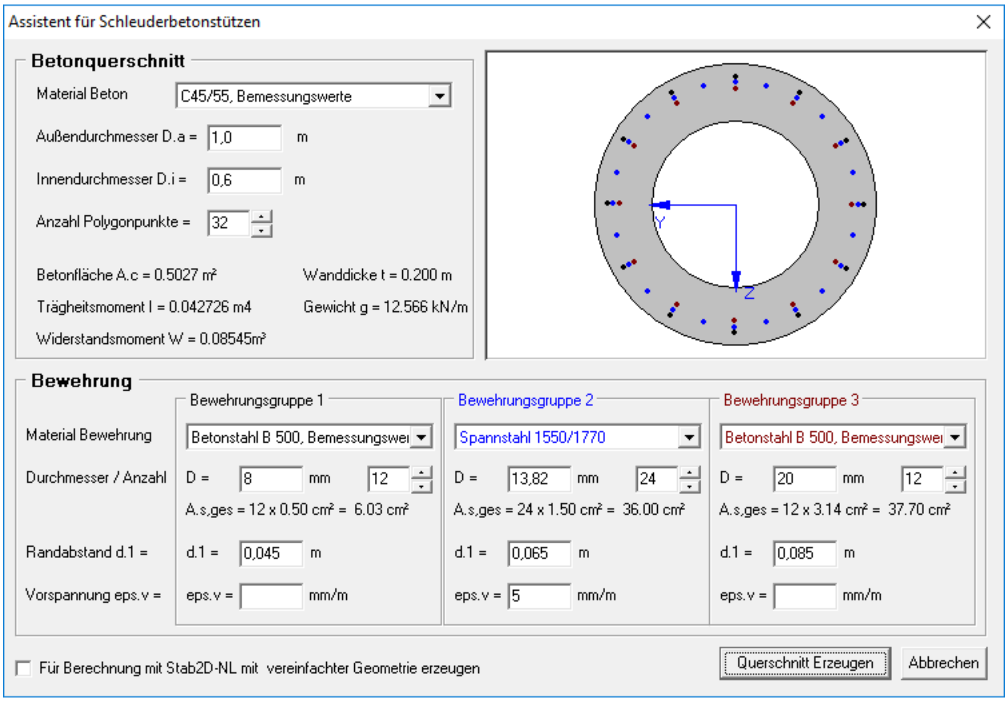
Use this tab to create a circular cross-section, optionally a circular ring cross-section. The cross-section is defined as an equal-area polygon with 32 points. The reinforcement can be selected as a total area or by diameter.

Rectangle

Use this point to create rectangular or square hollow sections made of steel. The rounding of the corners of typical hollow steel sections is taken into account by the software and according to the user's specifications.

2.5.6 *Assistant for Spun Concrete Columns*

Menu ***Input***, sub-item ***Spun concrete columns***, the following window opens:



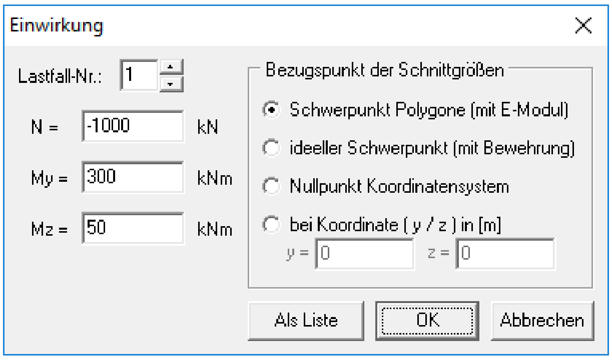
In this window, you can create spun concrete columns quickly and easily. Input values are:

- Outer and inner diameter
- Specifications for up to 3 different types of reinforcement in 3 layers, including specification of the prestressing of steel

In the graphic, the 3 types of reinforcement are shown in 3 different colors each.

2.6 Loads

Menu ***Input***, sub-item ***Loads***, the following windows opens:



Select the number of the load case and enter the values of the loads (axial force N_x, moment M_y and M_z). Select the point to which the forces are to be related (only important for axial forces). Axial compressive forces are defined negatively, tensile forces are positive.

As a matter of principle, INCA2 does not take into account any (additional) safety or combination coefficients within the program. Therefore please enter the loads in INCA2 as follows including the required factors:

- Ultimate Limit State (ULS) for load-bearing capacity – loads as design values (N_{xd}, M_{yd}, M_{zd}), i.e. including the corresponding partial safety factor and, if applicable, including combination coefficients
- Serviceability Limit State (SLS) – loads as characteristic values (N_{xk}, M_{yk}, M_{zk}), if necessary incl. combination coefficients

If you have selected user defined coordinates as a reference point, it is possible simply to move this reference point with the mouse using "drag and drop". This is particularly useful in demonstrations to explain the eccentricity of an axial force and the consequent changes in resulting actions or curvatures. In this way, for example, individual points of the moment/curvature relationship can be calculated:

$M = e \cdot N$ and

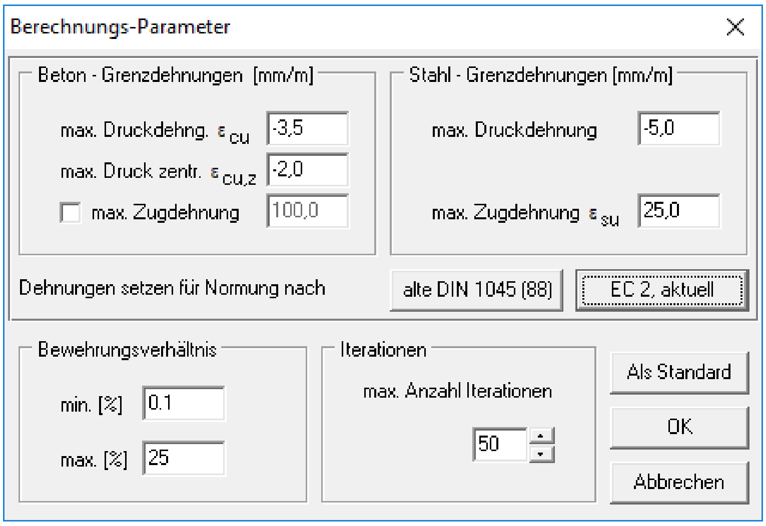
$1 / r = \Delta\varepsilon / \Delta\zeta$ (from the numerical results).

You can move the reference point by selecting the ***Move (Individually)*** button, so nothing else must be selected.

If you choose the ideal centre of gravity as the reference point, you will receive a corresponding result, depending on whether you have marked the point in the menu item ***Extras*** => sub-item ***Settings*** => tab ***Results*** that recesses in the concrete behind the reinforcement bar should be taken into account or not (***gross / net area of the concrete***)

2.7 Limit Strain / Parameters

Menu ***Input***, sub-item ***Limit Strains / Parameters***, the following window opens:



In this window, you can specify the limit strains of the cross-section that are needed to calculate the maximum bearing capacity. It is possible to use predefined limit strains for the ultimate limit state in accordance with Eurocode or to freely determine the strains by yourself.

Strains must be specified for both the concrete and the steel. The tensile strain of concrete is rather of minor importance and should be determined in such a way that this value never becomes decisive. Only in the case of a cross-section, where the reinforcement lies exactly on the edge of concrete-polygon, does this input prevent an impossible state of strain condition (curvature would be infinite). For more information, please also read the ***Chapter 6.8***.

In the lower part of the window you have the option to enter the reinforcement ratio and the number of iterations for a cross-section calculation.

For normal, simple cross-sections, 5 to 7 iterations are often sufficient for a convergent result. However, in the case of subsequently added cross-sections, more calculations are quickly necessary. The number of iterations should therefore be greater than 30. However, if equilibrium is still not reached after 50 to 100 iterations, then there will be probably no stable result. So 100 is a reasonable upper limit, the value 50 has proven to be sufficient for most cases.

The reinforcement ratio is important for the design of a cross-section and prevents over- or under-reinforcement. These values are used in the design as starting values for the iteration. It is checked whether the safety for the minimum reinforcement ratio is already given or whether no safety can be achieved even with the maximum reinforcement ratio.

Note for pre-stressed cross-sections:

With the procedure explained for the automatic design of reinforcement by INCA2, it can happen that the message of insufficient load-bearing capacity appears when designing a prestressed reinforcement. For the maximum reinforcement ratio, the cross-section was thus prestressed to such an extent that the prestressing overloaded the concrete cross-section. In such a case, it may make sense to reduce the maximum reinforcement ratio slightly.

2.8 Title/description for the system

In the last item of the ***Input menu***, you can enter a short text (maximum 65 characters) for the system, which will be output when printing, for example.

You will also find information about the system size, such as the number of points, cross-sections and used materials.

2.9 Enter pre-stress / pre-curvature

When entering polygons or reinforcing bars, it is always possible to define a pre-stress or pre-curvature for the corresponding cross-sectional part.

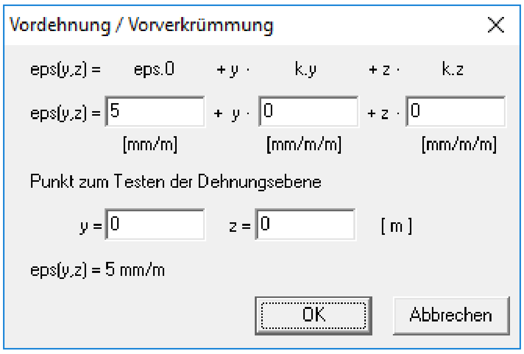
Areas of application of a pre-stressing):

- Pre-stress occurs, for example, in prestressed concrete. The stress in the prestressing steel-tendons can be applied to the prestressing strands as equivalent pre-expansion (in this case tension).
- For the shrink of concrete, the concrete shortens evenly, but is prevented from doing so by the reinforcement. As a result, a tensile stress builds up in the concrete and a compressive stress in the reinforcement. The modelling can therefore be carried out either by means of a tensile pre-expansion of the concrete or by a compression pre-expansion of the reinforcement.

Areas of application of a pre-curvature:

A pre-curvature (+ pre-stress) occurs, for example, in the case of precast concrete beams (longer beams), where concrete is added later and where the beam is without intermediate support. Due to the load from fresh concrete, the precast beam bends and already receives stress, but the fresh concrete is still stress-free after hardening. It is only through a further load (expansion loads, traffic) that this new concrete also participates in the transfer of the loads.

The pre-elongation + pre-curvature is entered as the strain plane in the following window:

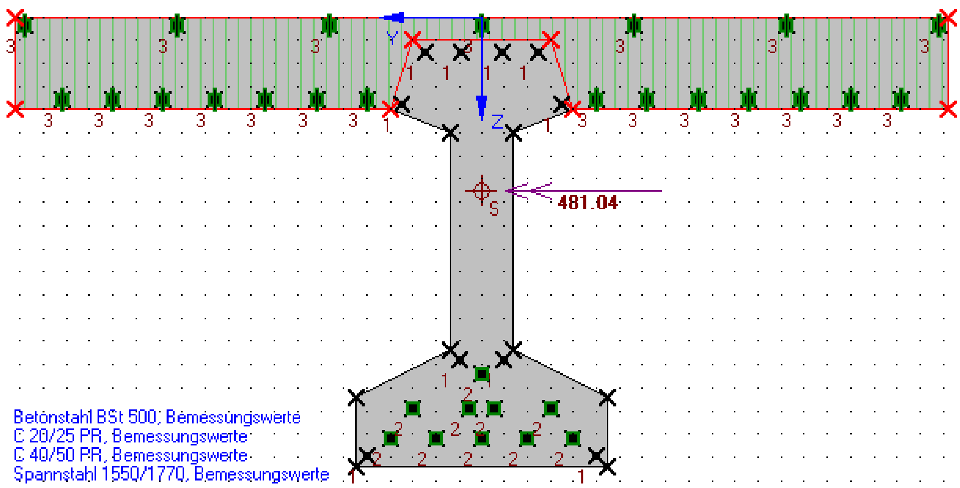


In the upper three text fields, enter the strain plane that you received in the case of a subsequently supplemented cross-section from an initial calculation of the precast element under concreting load. It is important that you swap the signs, so that "current strain plane" + "expansion plane from pre-expansion" result in zero in total and thus the additional concrete for this load remains stress-free.

To check whether all values have been entered correctly, a point can be entered as a test. In this fiber, the elongation (strain) is then calculated for control purposes.

For more information on pre-stretching, especially in the case of the interaction of concrete with tension in the cracked concrete tensile zone, see chapters ***8.3 Example 3 – Subsequently added bridge cross-section*** .

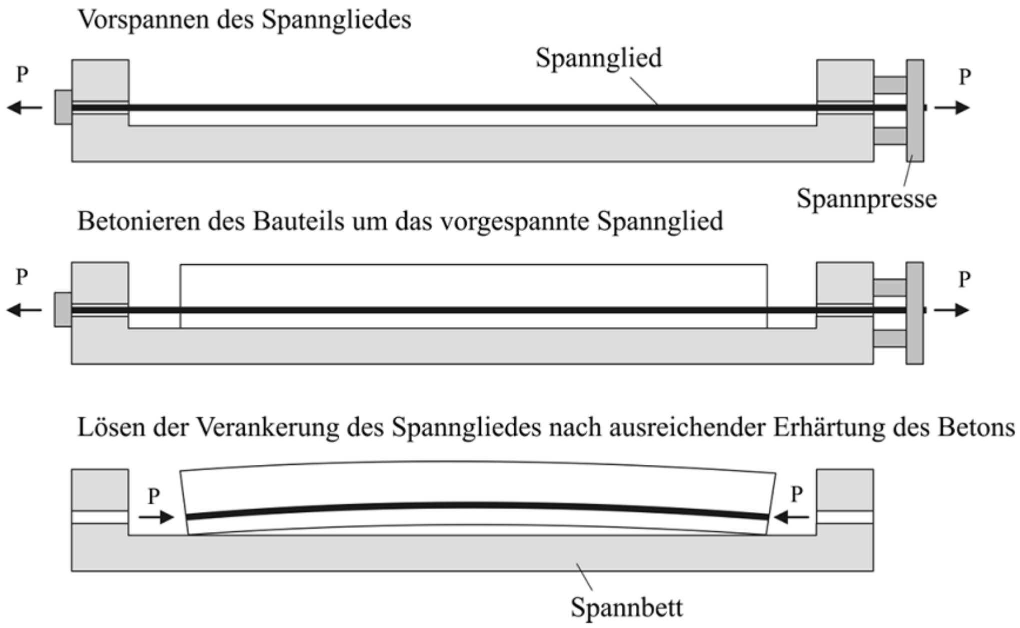
In the graphical displaying of the cross-section, pre-stretched reinforcement points are marked with a small, green square, while pre-curvatures are marked with small lines in the corresponding direction. Polygons with pre-expansion / pre-curvature are given a green border as well as a green hatch in the respective direction of the curvature.



Notes on the modelling of prestressing steel reinforcement

When defining the prestressing steel reinforcement, a pre-elongation of e.g. ε_p = 5.0 mm/m is entered. After calculating the cross-section without external loading, the elongation is reduced by the amount of concrete compression. Depending on the position in the cross-section, this results in residual prestressing steel strains of e.g. 4.7 mm/m. The difference, in this case 5.0 – 4.7 = 0.3 mm/m, is the measure by which the concrete was compressed. This modelling corresponds exactly to the **pre-stress in the clamping bed.** In a first step, the prestressing steel is installed in the tension bed and prestressed, then the concrete is poured stress-free. After the concrete has hardened, the prestressing steel is separated from the tension bed and the concrete is compressed. In this process, the original elongation of the prestressing steel is reduced by the extent of the compressive shortening of the concrete.

Prestressing in the clamping bed



Note for prestressing with <u>subsequent bonding</u> :

In the case of post-tensioning with subsequent bonding, the structural engineer defines a prestressing to be applied. This prestressing is applied directly by a tensioning press on the construction site and the concrete is pressed at the same time as it is tensioned. This means that the prestressing applied on the construction site already corresponds to the value resulting from the cross-section in equilibrium.

In INCA2, therefore, a prestressing must be defined that corresponds to the desired prestressing plus the concrete compression. In the case of simple cross-sections, this can basically be calculated manually beforehand. However, it is also possible to adjust the prestress iteratively, so that after calculating the strain state, the desired value of the preload is obtained.

Prestressing for a single-span beam

For the design of a prestressed single-span beam with a parabolic tendon path, determine the forces of dead weight and traffic with the respective partial safety coefficients. The internal forces resulting from the deflection forces of the tendon do not need to be determined, as the resulting stresses are already included in the INCA2 calculation with prestress.

You then perform the cross-section calculation with INCA2 and model the cross-section with the design values of the material properties for the concrete, reinforcing steel and prestressing steel. The tendon bars are pre-stressed accordingly. The forces of dead weight + traffic are applied and the checks are carried out for the various states in the ultimate limit state.

If verifications are to be carried out in the state of serviceability, the "mean values of the material properties" are to be selected for concrete, whereby the tensile strength is to be set to zero for the predefined material characteristics (calculation in the crack). This can then be used to carry out decompression checks, crack width checks or stress limitation checks.

2.10 Output of cross-section and input data

All output options can be found in the ***File*** menu, sub-item ***Output***.

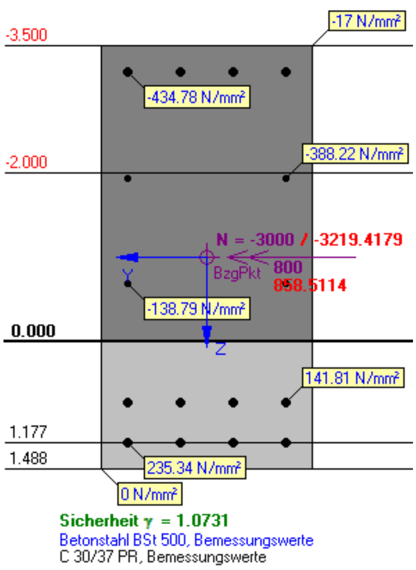
2.10.1 Cross-section printing

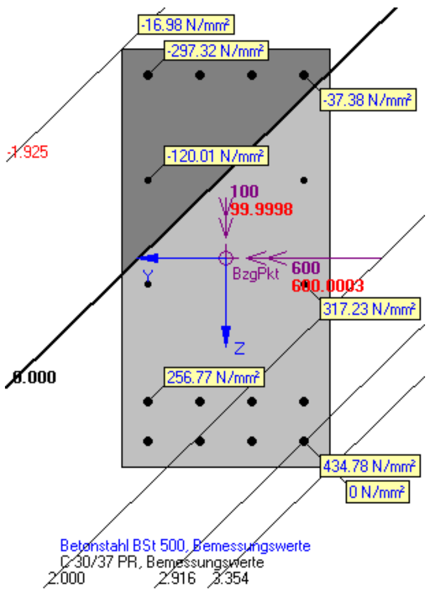
You can use this sub-item ***Cross-section Printing*** to plot the cross-section at a printer (shortcut ***Ctrl+P***). You have the ability to influence the printed image by adjusting the paper orientation, size, and positioning.

The cross-section will be adjusted to the size you have chosen. Grid dots and crosses at the polygon points are not printed. In addition, after a calculation has been performed, the numerical results can also be output during printing.

2.10.2 Graphic in clipboard

The cross-sectional image is copied to the clipboard as a pixel file and can then be pasted into Word etc. (shortcut ***Ctrl+C***). For an optimal result of the graphic in a static document, it has proven to be a good idea to reduce the displayed size of the cross-section a little and to arrange it in the lower left corner, where the used cross-sections are also displayed. If a safety check is carried out, the factor reaching the ultimate limit state is displayed at the bottom left.





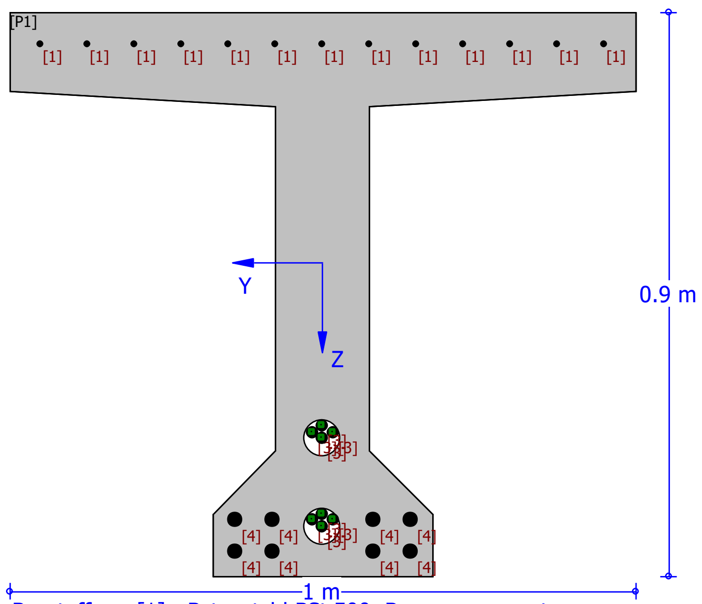
2.10.3 Save graphics as BMP

The cross-sectional graphic is saved as a pixel file in bitmap format (BMP).

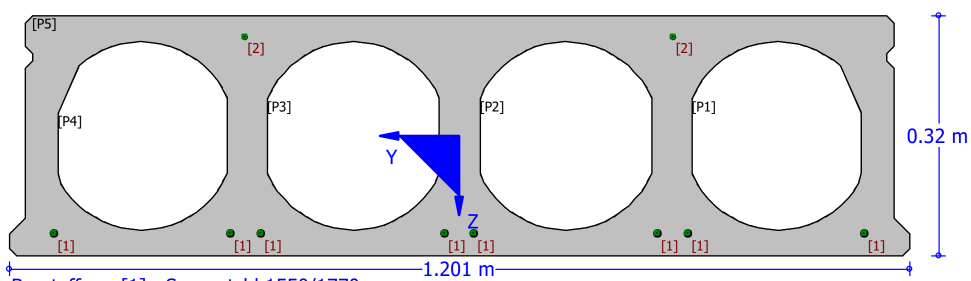
2.10.4 Input data graphic in clipboard

For quick documentation in static calculations, the following output can be helpful, in which the cross-section with main dimensions, the materials as well as the reinforcement quantities and reinforcement groups are displayed in a graphic. The graphic is exported to the clipboard as a metafile (vector graphic) using the ***Ctrl+D*** shortcut.

In the graphic, the different polygons are labeled with [P...] and the reinforcement points with the number of the reinforcement group. The outer edges are measured with the dimension chains. The areas of the polygons and the area sum of the reinforcement group are specified. Negative values are areas that have been defined as recesses.



Baustoffe: [1] - Betonstahl BSt 500, Bemessungswerte
[2] - Spannstahl 1550/1770, Bemessungswerte
[3] - C 35/45 PR, Bemessungswerte EC2
Bewehrung: [1] - A.s = 14.7 cm², Mat-Nr. 1
[2] - A.s = -56.55 cm², Mat-Nr. 3
[3] - A.s = 25.13 cm², Mat-Nr. 2
[4] - A.s = 39.27 cm², Mat-Nr. 1
Polygone: [1] - A = 0.2819 m², Mat-Nr. 3
A.ges = 0.2819 m² / A.s,ges = 79.11 cm² / roh = 2.81 %



Baustoffe: [1] - Spannstahl 1550/1770
[2] - C45/55, Bemessungswerte
Bewehrung: [1] - A.s = 9.82 cm², Mat-Nr. 1
[2] - A.s = 1.36 cm², Mat-Nr. 1
Polygone: [1] - A = -0.0469 m², Mat-Nr. 2
[2] - A = -0.0477 m², Mat-Nr. 2
[3] - A = -0.0477 m², Mat-Nr. 2
[4] - A = -0.0469 m², Mat-Nr. 2
[5] - A = 0.3722 m², Mat-Nr. 2
A.ges = 0.1831 m² / A.s,ges = 11.18 cm² / roh = 0.61 %

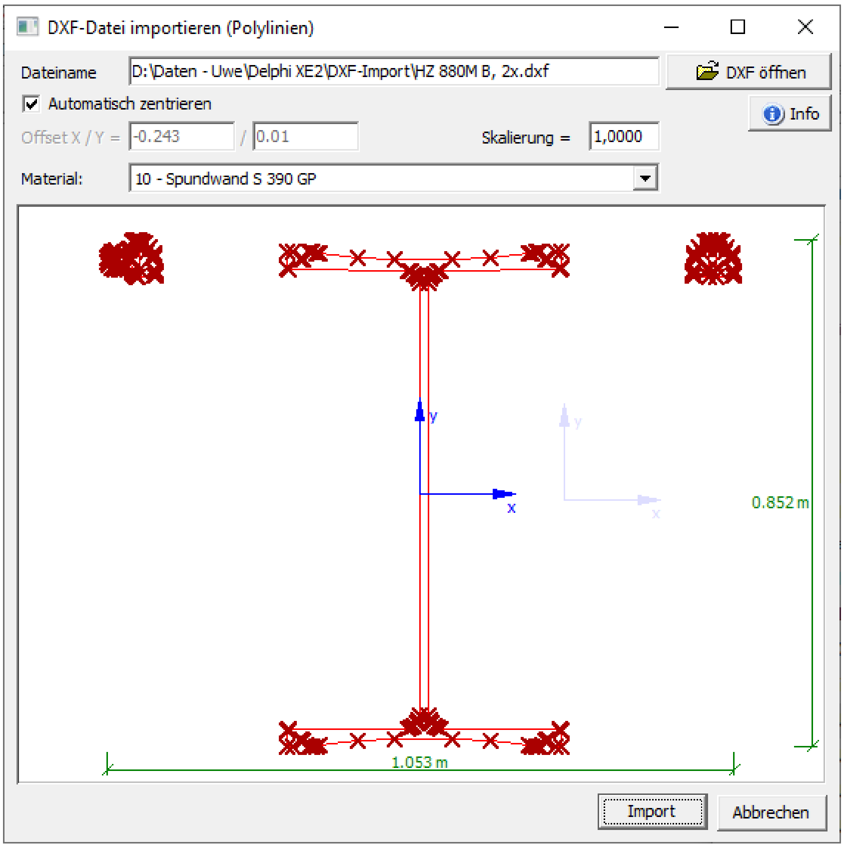
2.10.5 Input data (numeric + graphic)

With this sub-item, all input data (points, polygons, materials, etc.) are prepared as text and displayed in a separate window. The text can then be printed or copied.

Furthermore, the graphic of the cross-section is displayed in an extra window as a vector graphic. The graph corresponds to the output from the previous chapter 2.10.4.

2.11 Import DXF file

DXF files can be imported via the menu item *FILE* => *IMPORT DXF*. Only polylines that are generated as polygons in INCA2 are imported. The following window opens, in which a preview of the DXF file is displayed. In addition, values for the offset (moving the coordinate origin) and a scaling factor can be specified here.



When creating DXF files, keep in mind that only **polylines** are used. Other lines, blocks, or similar are not taken into account. You can use the PEDIT command in AutoCAD to join individual lines into a polyline. There are slight differences between the different DXF versions that are not detected by INCA2. Therefore, after reading the file, check whether all elements have been transferred correctly.

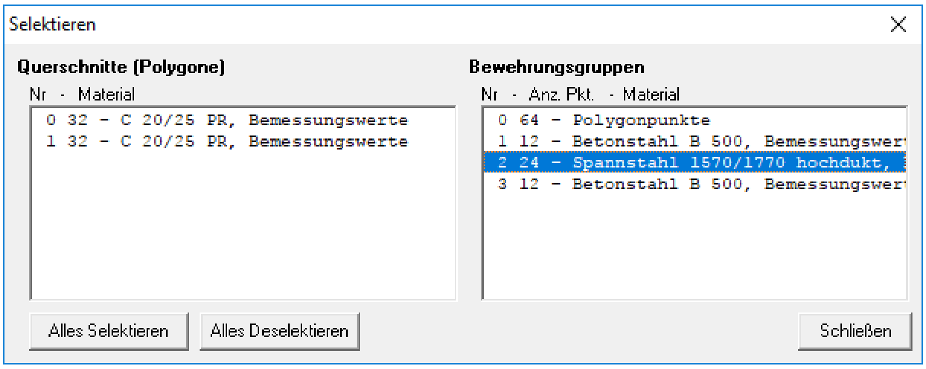
3 Edit

3.1 Select

The selection is made with the mouse by simply clicking on the point or the polygon. It is also possible to drag a window, with all elements that are completely inside the window highlighted in red. By pressing the Ctrl key, multiple selection is possible, as it is usual in other software. De-marking with the Ctrl key is also available.

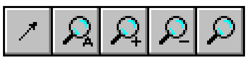
If you have selected elements, the functions Slide, Flip and Rotate as well as Properties are available. If no elements are selected, only the function of moving individual items can be activated.

In the Edit, Select menu, a tool is available for selecting entire reinforcement groups or individual polygons. This is helpful to get an overview of the individual groups, on the other hand, the reinforcement diameter of an entire group can be changed quite quickly.



3.2 Zoom Functions

The following zoom functions are available in the button bar:



1. ***Move View*** - The current section of the drawing is moved
2. ***Zoom All*** - The drawing is displayed in full format (window-filling) and centered.
3. ***Zoom in*** - The current displayed section is enlarged by a factor of 1.25.
4. ***Zoom out*** - The current displayed section is reduced by a factor of 0.8.
5. ***Zoom window*** - The mouse can be used to drag any window whose content is displayed in full format.

You can also move the displayed ***view*** by clicking the scroll wheel of the mouse, moving the mouse to move the view, and releasing the mouse button.

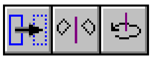
The ***zoom in*** and ***zoom out*** functions can also be done by turning the scroll wheel of the mouse.

3.3 Sliding, mirroring, rotating

If no elements are selected, the ***Move Individual Elements*** function is available. With it, especially in interactive mode, it is possible to quickly move reinforcing bars or polygon points via "drag and drop" (click, hold and move element, release the mouse button at the destination). It is also possible to move the force application point if any reference point has been selected when entering the load case.

 Move individual elements

Once elements have been selected, the following functions are available:



The buttons have the following meanings (from left to right)

1. Slide / Move
2. Mirror
3. Rotate

In a first step, select points and/or polygons and then select the desired action using the corresponding button or the ***Edit menu***.

If you select from the ***Edit*** menu or double-click on the corresponding button, a window will appear in which you can carry out the action using keyboard input. If you want to create new points / polygons, select the ***Copy option***. When pushing, you can also enter the ***number of copies***. The angle of rotation is used in the mathematically positive sense (counterclockwise).

If you prefer to use the mouse, click on the corresponding button in the input mask (e.g. ***Slide with the mouse***), the window disappears and the mouse cursor changes according to your selection. Click on a starting point in the graphic, hold down the mouse button and release it at the target point. In the lower status bar of the main window, the absolute and relative coordinates are displayed as an aid.

Clicking on the right mouse button will take you back to the input mask and cancel the function.

When using the mouse, the new position of the points is constantly displayed on the screen.

3.4 Delete

Mark the points and polygons you want to delete. Then press the ***Delete*** key or select ***Delete*** from the ***Edit menu***.

In the ***Tools*** menu, ***Settings***, ***Edit*** tab, you can specify whether a confirmation should be obtained before deleting. You can also specify whether you want to remove the associated points when you delete a polygon.

3.5 Properties

With this point, you can display the properties of points and polygons and also change them. To do this, select the desired elements and then select ***Properties*** (or ***Properties*** button or ***Alt + Enter***) from the ***Edit*** menu.

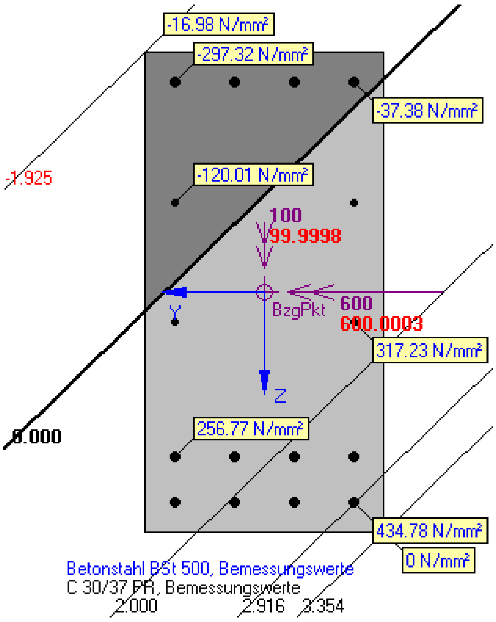
If you have selected several points or polygons, only the common properties are displayed, the other fields remain blank but can be changed. (e.g. if all points are made of the same material or if they have the same coordinates.)

4 Results

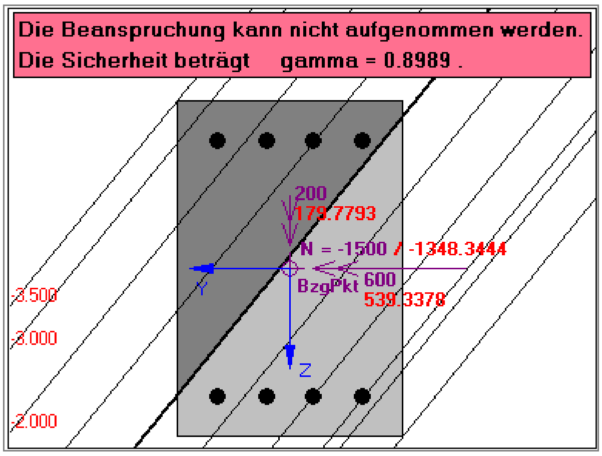
4.1 Calculation of strains and stresses (interactive)

After entering a cross-section and a load, you can calculate and display the strain distribution for the cross-section by selecting this menu item (shortcut *F9*).

The compression zone of the cross-section is filled in dark grey, the tensile zone light grey. At the pre-selected distance (menu Extras, sub-item Settings, Results) the lines of equal strain are displayed. The most stressed fibres for concrete and reinforcement are additionally marked with a line. Optionally, the stresses can be output in the various cross-sectional parts. The load acting in this load case is shown at the chosen reference point.



If this load is too large, the maximum acceptable load is shown here. In this case, the max. bearing capacity of the section is calculated, the corresponding safety factor is shown in the upper left red field.



The second sub-item ***Strain – All Load Cases*** (Shortcut ***Ctrl+F9***) is primarily intended for daily work in the engineering office. For example, in fatigue tests or generally in the case of serviceability verifications, stresses for different load cases often have to be determined. With this menu item, the strain states are determined for all load cases and the detailed numerical results are displayed in an extra window. For the current load case, the result with strain state is displayed.

In particular, the third sub-item ***Stretching - Interactive*** offers great advantages for designing and constructing as well as for demonstrating and explaining in a lecture. If you have selected this point, the strain distribution is automatically recalculated every time the cross-section changes. For example, you can change the position of the reinforcement and immediately see the effect on the cross-section and, if necessary, the maximum load that can be taken.

By selecting the sub-item ***Stretch - Interactive*** again , this mode will be switched off again.

If errors occur during the calculation, the result output is cancelled and, if possible, the type of error is explained. On the one hand, this can be caused by the non-convergence of the iteration, which, however, is difficult for the user to remedy. The most common reason, however, is an error in the definition of the material properties, which should be checked again in such a case.

If the interactive mode of strain calculation was activated, this mode is usually switched off by the software in the event of errors.

4.2 Output of results

In the ***File*** menu, sub-item ***Print Graphics***, you can print the cross-section. When printing, you have the option of printing the (short) numerical results together with the graphic on one page by checking the appropriate box.

The numerical results (as a short summary) are called up in the menu ***Results***, sub-item ***Numerical results*** (shortcut N). The results are displayed in a window in the upper right corner, which is always updated for further calculations and always stay in the foreground. In this window, you can set the scope of the numerical result in the menu item ***Display*** (selection: ***Stresses*** / ***Legend*** / ***Material Parameters***).

Example of numerical results for calculation of strain condition:

```
Elongation Condition for Given Internal Forces

 N : -1000.0000 M.y : 300.0000 M.z : 50.0000     Input
 N = -1000.0004    M.y = 300.0000 M.z = 50.0000  from iteration
 Internal forces were related to the point ( 0.0000/ 0.0000).

 alpha.0 = 118.2716 y.0 = -0.0490 z.0 = 0.0264

 eps.0 = -0.0680 deps/dy = -1.3873 deps/dz = 2.5796
 eps.2b = -1.1193 eps.1b = 0.9834 eps.1s = 0.7057
```

In the first result line with the forces, the user defined forces are shown. The second line shows the forces resulting from the iteration and stress integration. These forces may differ slightly from the entered forces due to the abort accuracy during iteration.

Explanations of the individual values can be found below under the keyword ***Legend***.

If in the ***Display*** menu the sub-item ***Stresses*** is selected, the maximum and minimum stresses are displayed, separately for all polygons and for all reinforcement groups.

```
Stresses in cross-section in [N/mm²]:
 Polygon 1 Sigma.Max = 0.0000 Sigma.Min = -15.9877 (C 35/45 PR, rated values)
 Bew-Gr.  1 Sigma.Max = 141.1343 Sigma.Min = -168.3287 (reinforcing steel B 500, design val-
ues)
```

If you have selected the sub-item ***Material Characteristics*** in the ***Display*** menu, the material parameters, e.g. for the documentation in a structural analysis, are also displayed.

```
Reinforcing steel B 500, design values, steel, parabolic rectangle
   E = 200000 N/mm²
   Pressure: Sigma.y = -434.78 N/mm² / Eps.y = -2.174 mm/m / Exp = 1
   Pull: Sigma.y = 434.78 N/mm² / Eps.y = 2.174 mm/m / Exp = 1

C 35/45 PR, Design values, Concrete, Parabolic rectangle
   E = 19833.3 N/mm²
   Pressure: Sigma.y = -19.83 N/mm² / Eps.y = -2 mm/m / Exp = 2
   Pull: Sigma.y = 0 N/mm² / Eps.y = 0 mm/m / Exp = 2
```

If you select the ***Legend*** sub-item from the ***Display*** menu, the following explanations of the different names are given at the end of the numerical results:

```
Legend of numerical results:
-----------------------------------------------------------------------------
   alpha.0 = angle of the direction of curvature
   y.0 / z.0 = intersections of the strain zero line with the coordinate axes
   eps.0 = elongation at coordinate origin
   deps/dy, deps/dz = curvature in y- and z-direction
   eps.2b = concrete elongation on the pressure side
   eps.1b = concrete elongation on the tensile side
   eps.1s = steel elongation on the tensile side
-----------------------------------------------------------------------------
```

Detailed numerical results can be found in the ***Results*** menu, sub-item ***Detailed num. Results*** (Numerical results detailed). Here, for example, the results are displayed for each point with strain and tension in an extra window.

Example of numerical results for safety case:

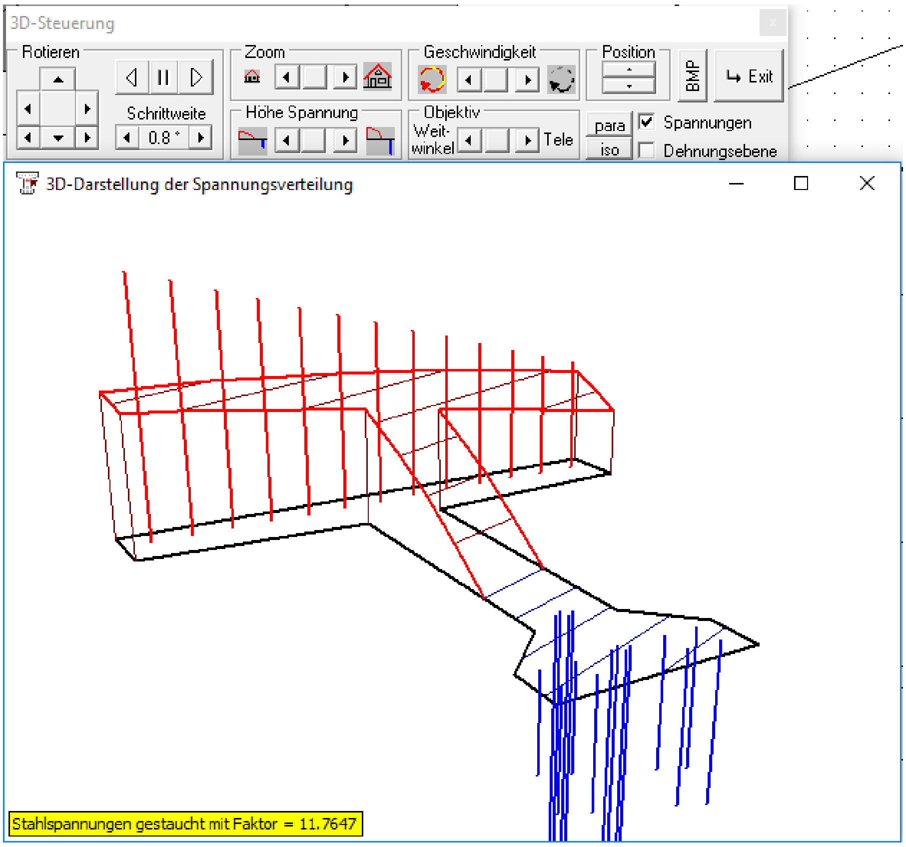
```
Safety Verification and Associated Elongation Condition
Safety factor until the limit state of the load-bearing capacity is reached
 gamma = 2.2053

 N : -1000.0000 M.y : 300.0000 M.z : 50.0000     Input
 N = -2205.3230 M.y = 661.5969 M.z = 110.2661    from calculation
 Internal forces were related to the point ( 0.0000/ 0.0000).

 alpha.0 = 124.3206 y.0 = -0.0639 z.0 = 0.0436

and so on.
```

The user entered the forces N_x = -1000 kN / My = 300 kNm / M_z = 50 kNm. This stress can be increased by a factor of γ = 2.2053 until the ultimate bearing capacity is reached, respectively the maximum strain at least at one cross-sectional fiber, which corresponds to the limit state of the cross-section. In this case, the bearing capacity in the ultimate limit state of the cross-section is N_x = -2205.3 kN / My = 661.6 kNm / M_z = 110.3 kNm.

The results can be deleted with the last sub-item of the menu Results, the graphic will be refreshed.

4.3 Stress distribution and strain plane as 3D animation

After successful calculation of the strain distribution, an animation of the stress distribution is possible. Calling up this sub-item opens two additional windows, a control panel and a viewport for the output of the 3D graphics.



In this window, there are buttons to control the rotation around the z-axis, and the object can be tilted towards the viewer and tilted to the side. With the directional cross on the left side, the 3D object can be rotated in 3° individual steps.

Furthermore, the height of the stress surface can be adjusted, and the zoom factor of the entire object can also be selected within certain limits. Position allows you to move the object vertically to fit it perfectly on the window. In addition, the appearance of the graphic can be influenced with the item Lens. With ***telephoto*** you get a parallel projection, with ***wide angle*** the object is displayed in central perspective (isometric).

Furthermore, you can use the buttons "***para***" and "***iso***" to select two views. In the first case, you get a view exactly from the side (90° to the strain lines), in addition, the lens is set to telephoto, which is comparable to a parallel perspective. With the button "***iso***" you switch to the isometric starting position.

With the ***BMP*** button you have the possibility to export the current graphic of the stress distribution as an image file in BMP format. With Exit, you leave the animation.

In addition to the control panel, you can rotate and tilt the graphic by clicking and dragging the mouse right/left and up/down in the graphic.

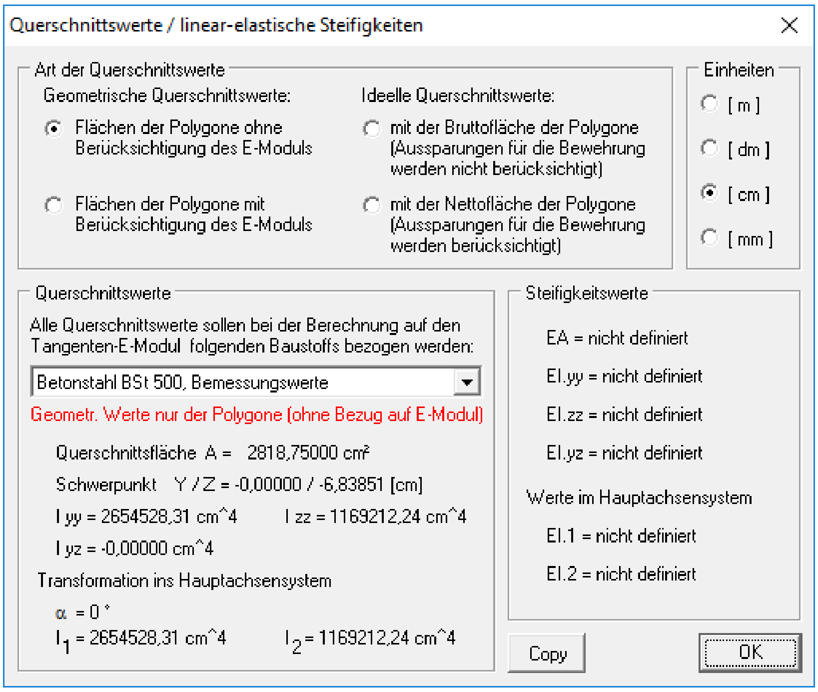
The previous diagram shows the stress distribution of a prestressed slab beam under biaxial bending. The pressure area is drawn in red, the tensile stresses in blue.

The quality of the displayed graphic of the stress in the concrete depends on the number and distance of the strain lines, especially in the parabolic region of the sigma-epsilon diagram (Tools menu, Settings sub-item, ***Results*** tab). The finer the subdivision by strain lines, the better the parabolic part can be depicted, e.g. in concrete. However, too many strain lines result in a high computational effort (slower animation), and the stress distribution becomes slightly confusing.

The steel stresses are usually compressed with a factor E.s / E.c (ratio of the modulus of elasticity). This measure allows a better representation of the concrete compressive stresses. In the graphics window, this compression factor is displayed at the bottom left.

4.4 Cross-sectional values

In this window, the current cross-section values such as area, centre of gravity and 2nd degree area moment are displayed. The latter is additionally transformed into the main axis system and the angle is calculated.



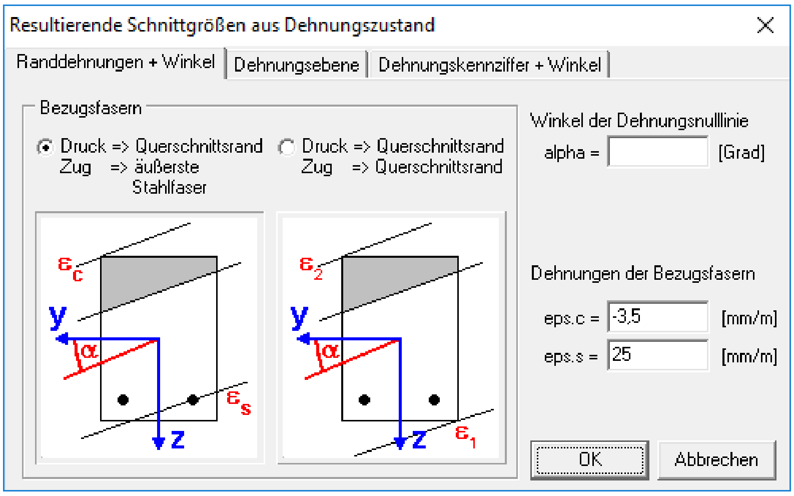
You can choose the calculation method as follows.

1. **Cross-section values only for the polygons**,
   reinforcement and modulus of elasticity are not taken into account
2. **Cross-section values only for polygons taking into account the modulus of elasticity,**
   reinforcement is not taken into account
3. **Ideal cross-sectional values (gross area),**
   consideration of the reinforcement and polygons with their respective modulus of elasticity, recesses in the concrete behind the reinforcement are not taken into account, the cross-section values are related to the modulus of elasticity of a material to be chosen by the user.
4. **Ideal cross-sectional values (net area),**
   consideration of the reinforcement and polygons with their respective modulus of elasticity, recesses in the concrete behind the reinforcement are taken into account, the cross-section values are related to the modulus of elasticity of a material to be chosen by the user.

In addition, the linear-elastic stiffness values (EA and EI) are displayed. Furthermore, there is a ***Copy*** button in this window, which is used to copy the geometric cross-section values to the clipboard. These can then be inserted into Word, for example.

4.5 Strain state and resulting stresses + forces

This point allows you to calculate the resulting internal forces from a given strain state (***Shortcut R***).



The strain distribution can be specified in 3 different ways:

1. Input of the strain on the compression and tension side. On the tension side, either related to the outermost concrete fibre or the outermost steel fibre. In addition, the angle of the strain zero line must be entered.

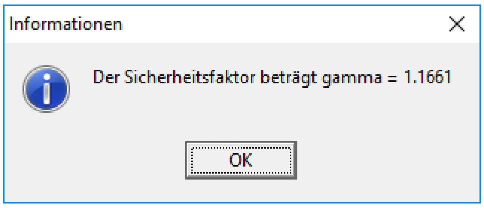
2. Input of a strain plane with the three parameters ε_0 (strain at coordinate origin), k_y and k_z (curvature in y- and z-direction)

3. Enter the DKZ, which is an internal value to code the strain-distribution. This value (0 to 33) clearly describes all possible limit strain states. As a result, the cross-section is in the limit state of load-bearing capacity. As with point 1, the angle of the strain zero line must be entered.

The result is the set of internal forces N_x / M_y / M_z, which result from the stress integration.

4.6 Safety factor

For the safety factor, the internal forces are calculated in the limit state of the load-bearing capacity, which result from the given internal forces by multiplying them by the safety factor γ to be determined (***shortcut S***).



The safety factor is displayed in a Message-Window, all other results can be found in the numerical results. If the output of the intermediate results (Tools menu, sub-item ***Settings***, Results tab, ***Iteration Progress***) tab is activated, the corresponding DKZ (strain code) can be read there.

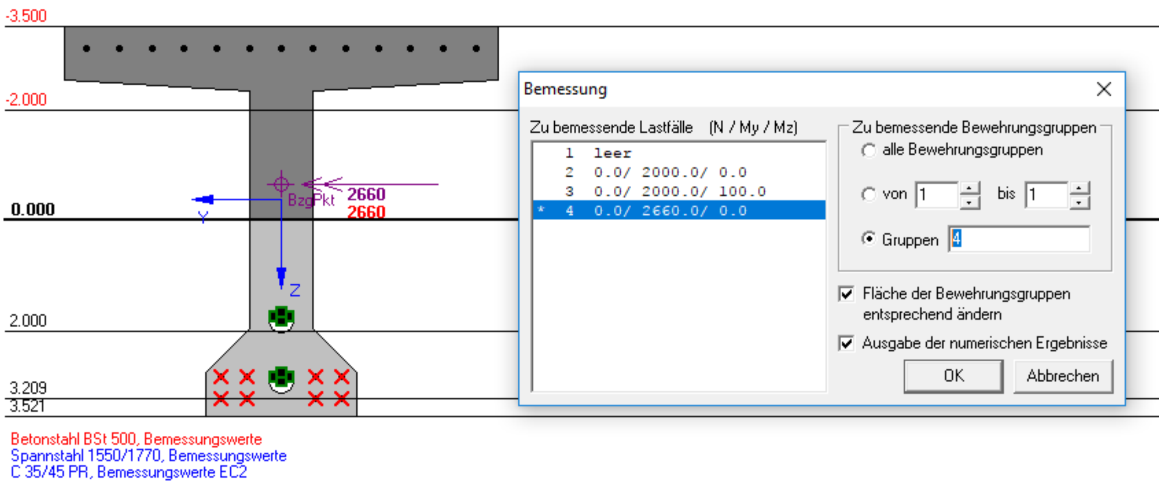
If the internal forces have already been entered as design values, the factor determined here is the additional safety between the user defined loads and the design load-bearing capacity of the cross-section.

Safety Case – All Load Cases

With this menu item, the safety check is carried out for all defined load cases and the numerical results are output in short form.

4.7 Design

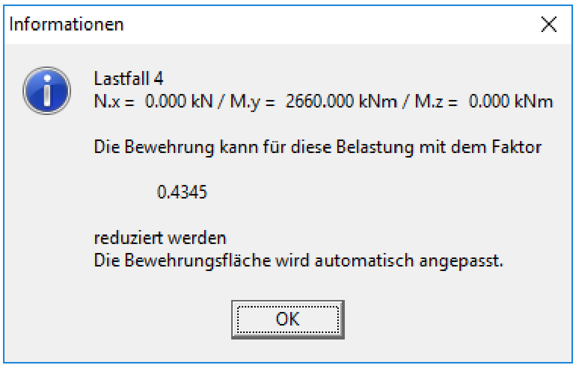
At this point, the cross-section is designed for a given combination of loads (***shortcut B***).



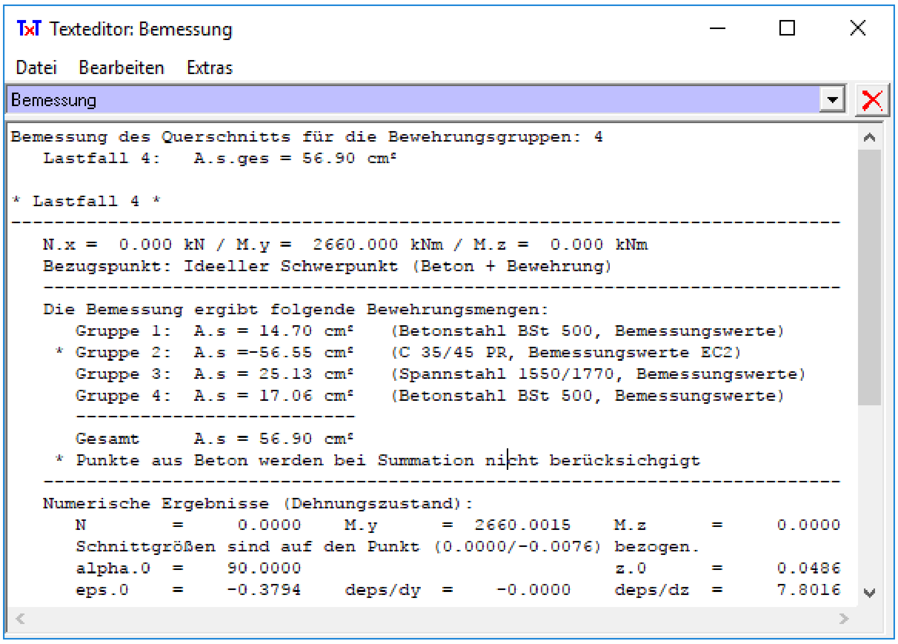
The amount of reinforcement of the specified reinforcement groups is varied until a safety factor of $\gamma = 1.0$ is reached. To do this, select the load case (or multiple load cases) for which the design shall be performed. In addition, it is necessary to select the reinforcement group that is to be adapted for the design. In the example shown, this is group 4 (slack reinforcing steel B500 on the bottom side of the cross-section).

For the design, the range of values is limited to the minimum and the maximum reinforcement ratio, which was entered in the menu item ***Input***, sub-item ***Limit Strains/Parameters***. If the safety of γ = 1.0 is not given even at the maximum reinforcement ratio, the safety check is carried out with this maximum reinforcement quantity. If, on the other hand, even the minimum amount of reinforcement is sufficient, the minimum reinforcement of the safety verifications is also used here and the value for γ is calculated.

If the calculation is successful, you will receive the following output:



Numerical results:




```
Bemessung des Querschnitts für die Bewehrungsgruppen: 4
   Lastfall 4:   A.s.ges = 56.90 cm²

* Lastfall 4 *
-----------------------------------------------------------------------------
   N.x =  0.000 kN / M.y =  2660.000 kNm / M.z =  0.000 kNm
   Bezugspunkt: Ideeller Schwerpunkt (Beton + Bewehrung)
   --------------------------------------------------------------------------
   Die Bemessung ergibt folgende Bewehrungsmengen:
      Gruppe 1:  A.s = 14.70 cm²   (Betonstahl BSt 500, Bemessungswerte)
    * Gruppe 2:  A.s =-56.55 cm²   (C 35/45 PR, Bemessungswerte EC2)
      Gruppe 3:  A.s = 25.13 cm²   (Spannstahl 1550/1770, Bemessungswerte)
      Gruppe 4:  A.s = 17.06 cm²   (Betonstahl BSt 500, Bemessungswerte)
      --------------------------
      Gesamt     A.s = 56.90 cm²
    * Punkte aus Beton werden bei Summation nicht berücksichgigt
   --------------------------------------------------------------------------
   Numerische Ergebnisse (Dehnungszustand):
      N         =     0.0000    M.y      =  2660.0015    M.z      =     0.0000
      Schnittgrößen sind auf den Punkt (0.0000/-0.0076) bezogen.
      alpha.0   =    90.0000                             z.0      =     0.0486
      eps.0     =    -0.3794    deps/dy  =    -0.0000    deps/dz  =     7.8016
```

Note:

With prestressed cross-sections, the automatic design routine for prestressed reinforcement groups may not work correctly. At the beginning, the load-bearing capacity is checked with the minimum and maximum reinforcement ratio. In the case of a very high prestressing reinforcement, this can lead to the result, that the cross section is over-loaded due to the high prestress alone, so that the program comes to the conclusion that a design is not possible.

In such a case, the prestressing reinforcement must be increased “by hand” until the load-bearing capacity is achieved. Apart from that, in most cases the prestressing reinforcement is selected according to the requirements in the serviceability state (fully compressed cross-section) and, if necessary, additional slack reinforcement is inserted for the ultimate limit state.

4.8 Interaction Graph N / My / M_z

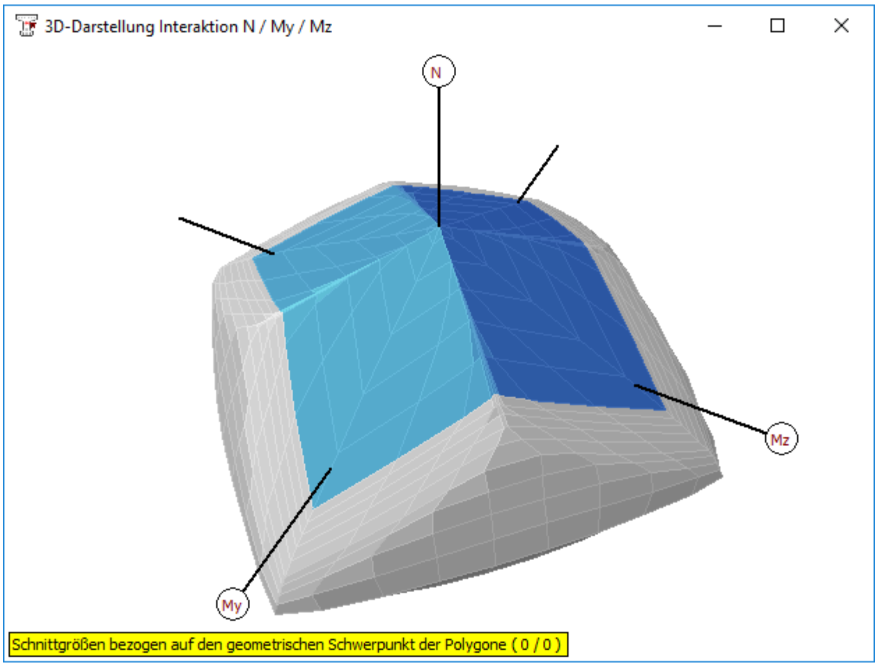
This menu item calculates the M/N line for all main curvature directions at an angular distance of 10°. Taken together, these lines form a 3-dimensional body, which reflects the internal forces of the cross-section as an interaction between N_x, M_y and M_z .

As with the 3D stress and strain distribution, a window for graphical output opens as well as a window for controlling the rotation, which has almost identical functions for sequence control.

In addition, there is a "light switch" here that can be used to turn the shading of the surface on and off. This shading enhances the 3-dimensional effect, but is more complex to calculate during rotation (and thus slightly slower).

In addition to the control panel, you can click and drag the mouse to rotate and tilt the graphic right/left and up/down

$N_x/M_y/M_z$ - Diagram for a double-symmetrical column cross-section:



The blue area represents the results of the strain ratios (DKZ) 33 to 16 (tensile fracture range), the red area the resultants of the strain ratios 6 to 0 (approximately overcompressed cross-section).

In earlier versions of INCA2 or MasQueW, the subdivision of the strain codes (DKZ) 0 to 6 was used for the range "centrically pressed" to "decompression" (edge strain on one side $\varepsilon = 0$). In the present version of INCA2, however, the definition of the DKZ had to be revised to adapt to composite cross-sections with external steel fibre, so that a clear assignment in this area is no longer possible, which is why the section DKZ from 0 to 6 only approximates the area of the completely overpressed cross-section.

For more detailed information on the old definition of the strain code for the description of the limit state of the load-bearing capacity, please read issue 415 DAfStb "Program-controlled calculation of arbitrary solid cross-sections under biaxial bending with longitudinal force (program MasQue)" by Busjaeger, Quast (1990). At the Institute of Concrete Structures (3-07) of the TUHH, a booklet with the same content is available for reimbursement of the cost of production.

Optimized spacing:

For this point, the respective M/N lines are determined with an optimized distribution of the calculated points. Due to the high limit strains in the new generation of standardization, especially for steel, some of the results do not change for DKZ = 20 to 33. The same may also apply to the DKZ = 0 to 1 interval. These intervals are therefore excluded from the calculation in order to calculate more intermediate points in the range of interest. The advantage is that the 3D body is represented with more calculated points and thus more precisely in shape. A disadvantage, however, is that the color assignment (tensile fracture and compression fracture area) is no longer correct.

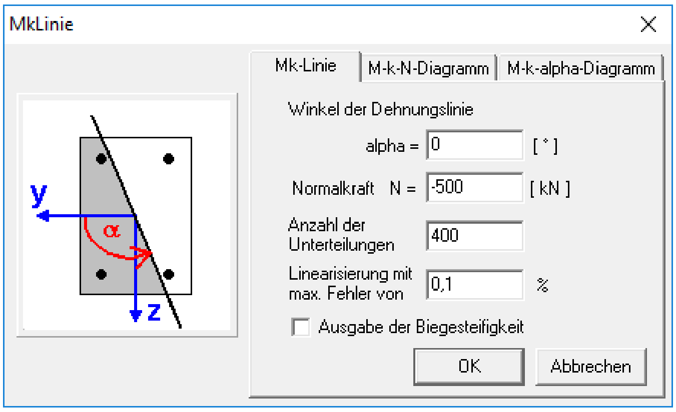
Distances such as DKZ:

This means that the calculation is carried out according to the strain indicators, with the disadvantage that at high limit strains for steel, the 3D body is represented with only a few points and thus less precisely in shape. On the other hand, the color-coded areas (red and blue) roughly delimit the tensile fracture and compression fracture areas.

4.9 M/K Line

This menu item calculates the moment curvature line of the cross-section (***shortcut M***). The input values are the angle of the zero strain line and the constant axial force. Furthermore, the number of subdivisions for the line must be entered. Since the buckling points of the M/k line (e.g. cracking of the cross-section or yielding of the reinforcement) are not recorded separately, a comparatively high subdivision is required to capture these points with good accuracy. A number of 400 subdivisions have proven their worth here.

If the M/k line is used in further calculations, it is recommended to reduce the number of points. For this purpose, successive pairs of values are checked for deviations from a section-by-section linearization. Points with a slight deviation are then omitted. To do this, enter the maximum error percentage in the last text box. An accuracy of 0.1% to 2% has proven to be effective to get such a simplified line.



The results are displayed once in graphical form in a separate window (metafile, vector graphics), as well as the numerical results, which are displayed in tabular form in a separate window with all important associated variables. In this table, the following values are displayed:

- EPS0 — Elongation of the cross-section at the origin of the coordinate system
- k.tot — Total curvature = $(ky^2 + kz^2)^{0,5}$
- N, M_y, M_z — Internal forces
- M.tot — Total Moment = $(My^2 + Mz^2)^{0,5}$ where the sign corresponds to the orientation of the total moment vector
- alphaM — Angle of Total Moment Vector
- Anz.Sp. — Number of stress integrations to determine the point on the M/k line. The sign indicates whether the start value of the iteration is greater than or less than the final value (just to check the iteration-procedure)

When determining the M/k line, the limit state (maximum positive and maximum negative curvature) is first calculated. In addition, the range between the boundary curvatures is divided into equal sections and the corresponding bending moment is determined for each curvature value. Unfortunately, this simple procedure does not capture the characteristic points of the M/k line, which are caused by the cracking or yielding of the reinforcement, for example. Until now, these points in the M/k line can only be made visible by subdividing them as finely as possible.

At the end of the numerical output of the Mk line, the linearized line is output for the software ABaS (static software from Prof. Quast, Nonlinear Calculations of 1D Beam Structures) as well as for Excel. Depending on the accuracy requirement, unimportant values of the lines are not displayed. An important reason for this is that ABaS can process a maximum of 50 pairs of values of the Mk line for a nonlinear deformation calculation. When calculating with ABaS, please also make sure that the Mk line does not have any falling areas (negative tangential bending stiffness). Such problems cannot be solved with conventional beam statics, as two different curvatures occur at a bending moment, meaning the assignment is no longer clear.

The tabs 2 and 3 provide further options for calculating Mk lines. The M-k-N diagram with 2D or 3D output can be helpful to study the influence of the axial force on the maximum bending moment or the achievable curvatures in more detail. The M-k-alpha diagram shows the correlation at bending with $M_y + M_z$ when the main direction of curvature changes in the selected interval.

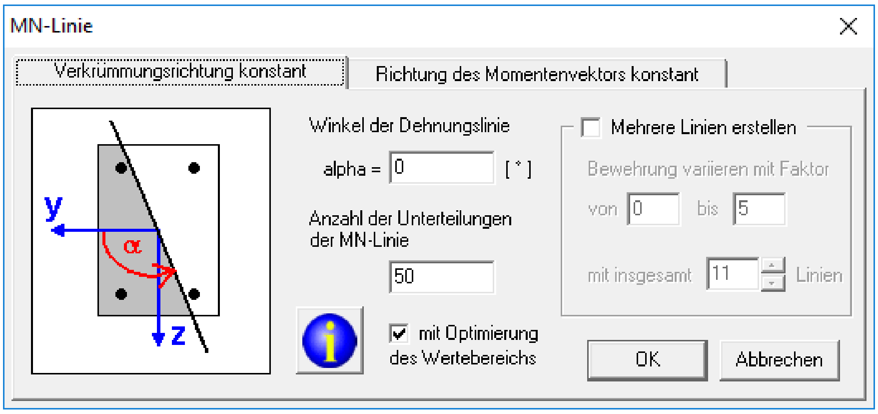
At this point, it should also be noted that when calculating the Mk line with the first two options, the main curvature direction alpha remains constant. Thus, the direction of the resulting moment vector will only be constant in certain cases and will have the same angle as the main curvature (e.g. alpha = 0° for single-symmetric cross-sections, alpha = 0° / 45° / 90° for double-symmetric cross-sections).

4.10 M/N Line

This menu item is used to calculate the moment-axial force line. The cross-section is in the limit state of the load-bearing capacity. The only input value is the angle of the strain zero line or the moment vector, and the number of subdivisions must also be selected.

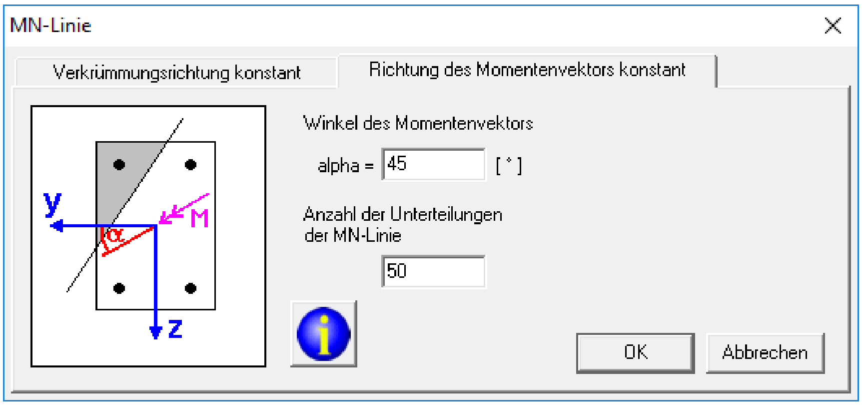
In the first tab-option, the MN line is calculated for a constant direction of curvature. In the case of asymmetrical cross-sections or a selected angle of the non-zero direction of curvature, the moment vector will have a changing angle depending on the strain state. Therefore, this M/N line generally does not represent a perpendicular section through the 3D-body of the N_x/My/M_z -interaction diagram. Only in special cases (in the case of uniaxial bending and symmetrical cross-sections) are the angles of the strain lines and the angle of the moment vector the same.

To determine this MN line, the range of the strain code (DKZ = 0 to 33) is divided into the selected number and the corresponding limit states of the cross-section are calculated. Characteristic points such as the cracking of the cross-section, yielding of the tensile or compressive reinforcement are not calculated separately.



Furthermore, it is possible to generate groups of MN lines in a single calculation process. To do this, select the option ***Creating Multiple Lines*** and enter the other values. This feature is useful for creating M/N design diagrams for special cross-sections that are not covered by the literature. However, it is also possible to create general interaction diagrams for other reinforcing bars (e.g. other strengths) (see example ***8.8 Interaction Diagrams Nx / My***).

As a second option, the MN line is calculated for a constant angle of the moment vector. Analogous to the first possibility, the angle of the main curvature or the zero line of strain now has a changing angle.



In this case, since the moment vector has a constant angle, this line represents a perpendicular section through the 3D-body of the N_x/My/M_z –interactiondiagram.

In both cases, the ***info button provides*** a brief guide to the type of calculation.

The results are displayed once in graphical form in a separate window (metafile, vector graphics), as well as the numerical results, which are displayed in tabular form in a separate window with all important associated variables. In this table, the following values are displayed:

- Strain plane with ε_0, k_y and k_z
- forces N_x, My, M_z
- Total bendiong moment M_{tot}
- Angle of total torque alphaM
- Strain index DKZ

Hint:

Due to the increased limit elongation for steel in the new standardization (according to DIN 1045-1 new or EC2: ε_{su} = 25 mm/m), there are no longer any differences in the results for the strain codes DKZ = 20 to 33 in some cases, because the concrete is cracked and the reinforcement is already completely in the yielding range. For this reason, before the calculation, it is checked in which area of the DKZ the resulting forces no longer change. In the subsequent calculation, the relevant area can then be displayed with more calculated points.

4.11 M_y / M_z-Line

This menu item is used to calculate the M_y-M_z line, which is practically a horizontal section through the 3D-body of N_x/My/M_z -interaction diagram. The cross-section is in the limit state of the load-bearing capacity. The only input value is the normal force, and the number of subdivisions must also be selected.

The results are displayed once in graphical form in a separate window (metafile, vector graphics), as well as the numerical results, which are displayed in tabular form in a separate window with all important associated variables. In this table, the following values are displayed:

- Strain plane with e_0, $k_{,y,}$ and k_z
- Internal forces N_x, M_y, M_z
- Strain index DKZ
- Angle of total curvature
- Number of stress integrations required in iteration

If the specified axial force is higher than the load-bearing capacity of the cross-section, a corresponding message is displayed indicating the maximum tensile and compressive axial force.

4.12 Deleting Results

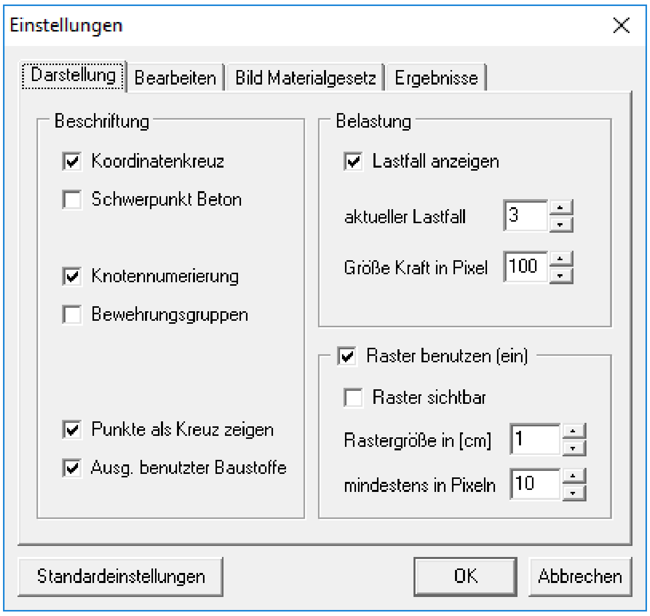
This menu item deletes the results of the strain calculation and closes the window for the numerical results (***shortcut Ctrl+L***). The extra windows for the output of the M/k lines etc. are not affected by this.

5 Menu Extras

5.1 Settings

If you select this menu item (***Tools*** menu, ***Settings*** sub-item), a window will appear with several tabs in which you can set the following items:

5.1.1 Graphic display



1. Coordinate cross – display on/off
2. Centre of gravity for concrete – display on / off
3. Node numbering – next to each node at the bottom right, starting at 0
4. Reinforcement Groups – Number of the reinforcement group next to each rebar in the lower left corner, starting at 1
5. Show points as cross – display on/off
6. Output of used materials – The materials currently used in the cross-section are displayed at the bottom left of the window. If a cross-sectional element is marked (e.g. reinforcing bar), the associated material is also shown in red at the bottom left.

7. Display Load Case – Display On/Off
8. Size Force in Pixels - Length of Moment Vector in graphic

9. Use a grid – a grid can be taken into account so that when working with the mouse (e.g. creating points, moving points, etc.), it is always rounded to "full" numerical values.
10. Grid visible – display on/off
11. Grid size - Setting the grid size in [cm] and in [screen pixel]

The grid serves as an aid when entering points or e.g. when moving. Points that you draw with the mouse are then generated with the coordinates of the grid points.

Use grid:
When drawing with the mouse, the grid points are used as coordinates

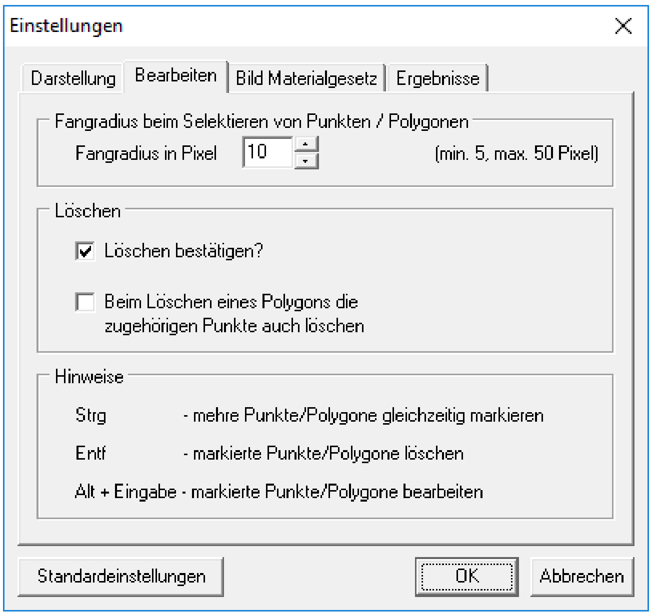
Use raster from:
The pixel coordinates of the mouse are converted into real coordinates of the system without rounding. In the display at the bottom left, only three decimal places are displayed, but the coordinates generated in this way are not rounded to 3 digits, but used unrounded.

Grid size in [cm]
If you want to create a system that has a cross-sectional point at the same distance, use this setting. However, the smallest pixel spacing of the grid is always set to 5 pixels, otherwise the recognizability suffers greatly.

Minimum size in pixels
The program will first try to display the grid size (in cm) you have chosen. If this is not possible due to the current zoom-factor, the coordinates are generated at multiples of tens of 1, 2 or 5. If the minimum distance of 10 pixels is selected, the actual distance between the grid dots can vary between 10 and approx. 24 pixels.

The narrowest grid is specified with at least 5 pixels, because at a smaller distance a good and fast drawing with the mouse is no longer useful.

5.1.2 Edit

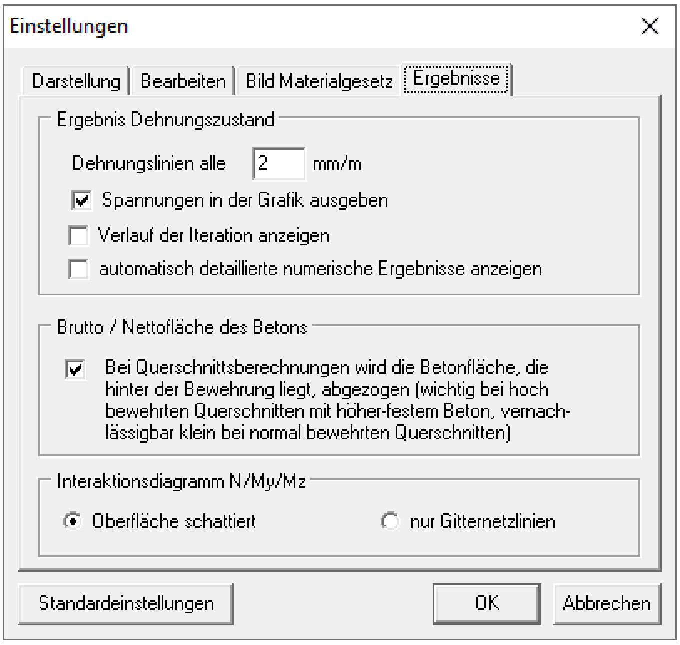


1. Snap Radius – Specifies the distance (in pixels) at which a point or polygon should be selected (selected) by mouse.
2. Confirm deletion – a security prompt is prompted before deleting points or polygons
3. When deleting a polygon, the associated points can also be deleted. Note that the points you want to delete are not also part of another polygon.

5.1.3 Picture of Material Behaviour

Here you can set colors, line thickness and window size (resolution) for the output of the material laws. These values are also used to display the M/k line, M/N line and My/Mz line.

5.1.4 Results



Strain Lines:

Here you choose whether and at what distance the strain lines are displayed in the graphic results. Regardless of this, the strain-zero line as well as the most stressed concrete and reinforcement fiber are always marked with a strain line.

In the case of results with large strains (e.g. in the limit state of the load-bearing capacity) and at the same time very small distances of the strain lines (e.g. 0.1 mm/m), so many strain lines can be displayed that the graph is no longer displayed correctly. In this case, the distance between the stretch lines should be increased.

This setting is also important for the quality of the animated 3D representation of the stress distribution over the cross-section, as the quality of the 3D stress model depends on it. The finer the subdivision by expansion lines, the better the parabolic part can be depicted, e.g. in concrete. However, too many strain lines result in a high computational effort (slower animation), and the stress distribution becomes slightly confusing.

Output strains in the Graph

After a calculation, the points with the maximum and minimum stress of each polygon and each reinforcement group are labeled.

Iteration progression:

In case of problems with the strain calculation (non-convergence in the strain state, safety case), you can set here whether the intermediate results should be output during the iteration. This gives a rough overview of where the problem of non-convergence might lie.

Numerical results

In this case, the detailed numerical results are automatically shown and updated after a calculation has been performed.

Gross/net area of concrete

Behind a defined reinforcement bar, there is usually still the concrete. At this point, you can choose whether this concrete area should be taken into account or subtracted. In order to check a calculation result by hand, it may be useful to calculate with the gross concrete area (no recess). However, especially for higher-reinforced cross-sections with high-strength concrete, the recesses for the reinforcement should no longer be neglected and a calculation with the net concrete area should be carried out. In the case of highly reinforced columns, the load-bearing capacity can otherwise quickly be overestimated by 10% or more.

In the detailed output of the numerical results, the reinforcement points for which a recess has been defined are marked with an asterisk.

Interaction Diagram:

Here you can set the display of the 3D interaction diagram. In the first case, the program's own routines are used. If OpenGL is chosen, either the operating systemroutines or the graphics card routines are used for display.

5.2 Measure

Select this menu item to determine the distance between two points or the angle with three points. After selecting one of these functions, the mouse cursor changes. You can cancel with the right mouse button.

Shortcuts	Ctrl + A	Measuring Distance
	Ctrl + W	Measuring Angles

5.3 Concrete net area

As explained above, the concrete is normally still behind a defined reinforcement point. In the menu ***Tools*** => sub-item ***Settings*** => tab Results tab, you can choose whether this concrete area should be taken into account or subtracted. However, this consideration is only taken into account internally during the calculation and is therefore only visible to the user in the output of the detailed results (where the reinforcing bars for which a recess in the concrete has been taken into account are marked with an asterisk *).

With the menu item ***Extras*** => ***Concrete net area*** presented here , the recesses in the INCA2 file can be created directly by creating further points with a negative surface and the material property of the underlying concrete polygon at the location of the reinforcement points. However, this menu item should only be used to check the exact results, as the reinforcement of the INCA2 cross-section is then somewhat more difficult to handle- With this method after all, there are two points on top of each other, so that selection is relatively difficult. User errors can also easily occur during design, as both the reinforcement group and the group with the associated recesses must be specified.

5.4 Overview Shortcuts

Shortcuts implemented in the program:

- Ctrl + Mouse Button Mark multiple points at the same time
- Delete Delete selected points / polygons
- Alt + Enter Edit Properties, Selected Points / Polygons
- F7 Open window for entering the loads
- F9 Calculating Strain Condition
- Ctrl + F9 Calculating Strain Condition for All Load Cases

- Ctrl + N New File
- Ctrl + O Open file
- Ctrl + S Save file
- Ctrl + P Print Graphic
- Ctrl + C Copy graphic to clipboard
- Ctrl + L Deleting Results
- Ctrl + E Open the Settings window “Extras”

- Ctrl + A Measuring Distance
- Ctrl + W Measuring Angles

- B Executing the design
- N Output of numerical results
- M Calculation of Mk lines
- S Safety calculation

6 Modelling of materials

6.1 Linear-elastic

Description:

- linear-elastic, Hooke's law fully valid

Application:

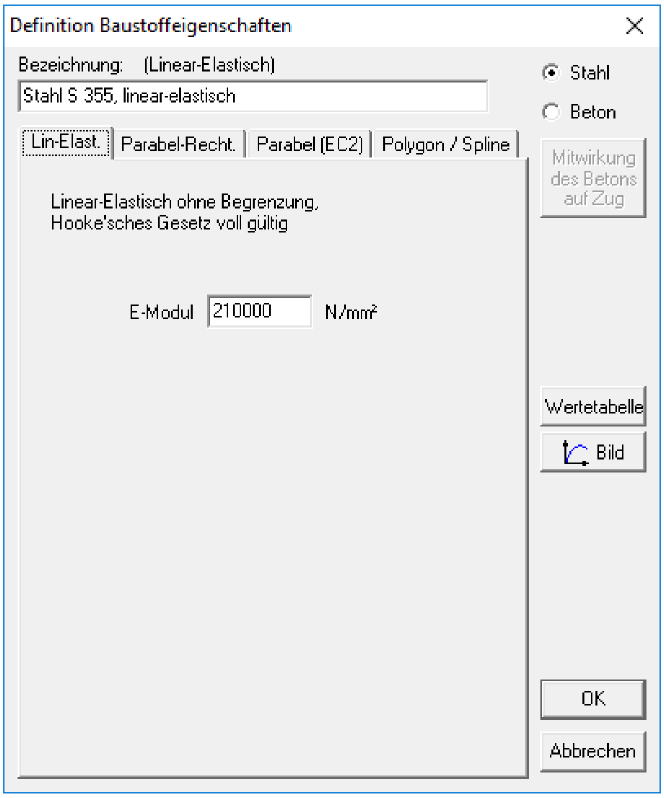
- linear-elastic calculations, e.g. steel in elastic state, without limitation

Advantage:

- Fast calculation of the strain condition without iteration

Disadvantage:

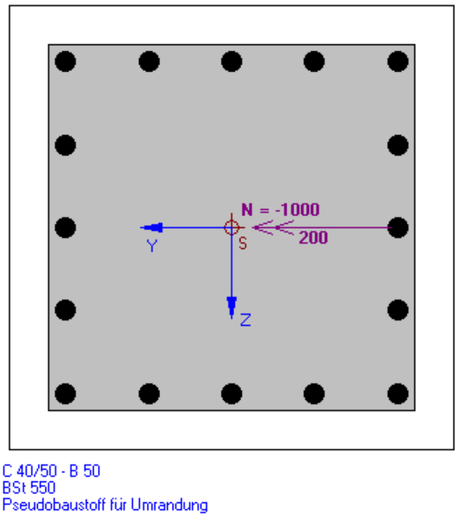
- only for simple calculations without material nonlinearities



The only input value for this material law is the modulus of elasticity.

Pseudo-material:

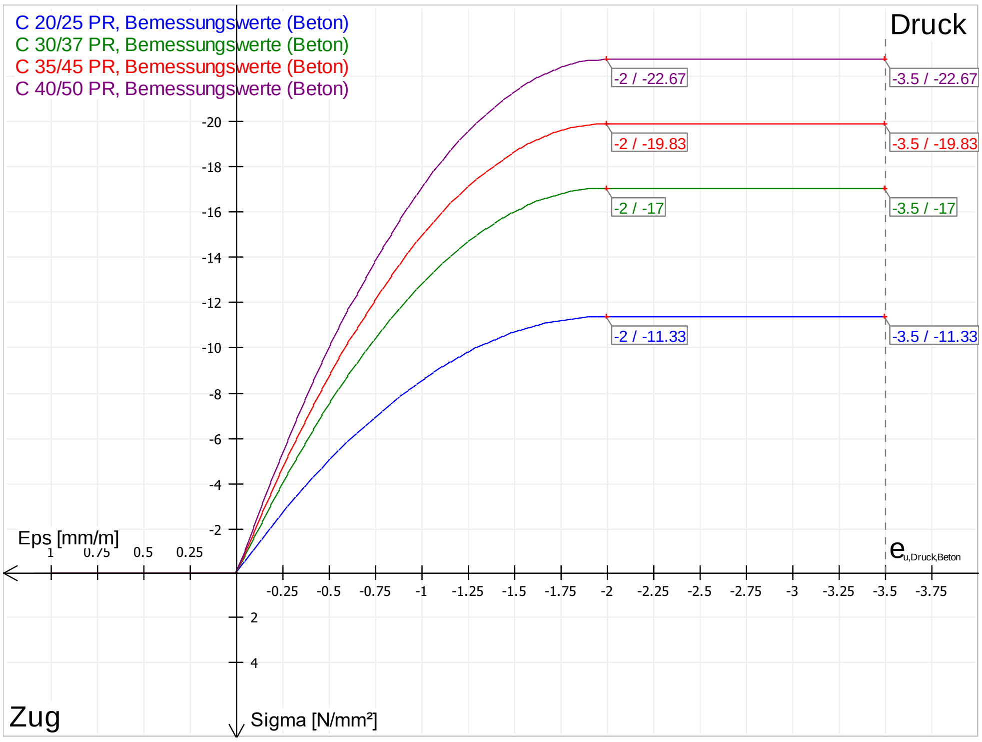
For some purposes, it may be useful to create a pseudo-material, which does not affect the calculation, just to clarify the cross section and give some additional information. The following picture shows a column with a reduced section for example during fire. The concrete cover up to the stirrup reinforcement is not taken into account in the verification in the limit state of the load-bearing capacity. Austrian standardisation provides for such safety aspect for highly reinforced columns, as tests have shown that the concrete cover of some columns flaked off shortly before reaching the maximum load. In order to represent the boundary in this case for better understanding, a pseudo-material can be defined with the modulus of elasticity E = 0. In order for this input not to be rejected by the program, the name of the material must contain the term "PSEUDO".



6.2 Parabolic Rectangle

Description:

- consists of a parabola in the first part and a linear second section
- Differing behaviour in the pressure and tension branch
- Participation of the concrete in the cracked concrete tensile zone according to the model by Quast and Espion possible, for calculation of deformations



Application:

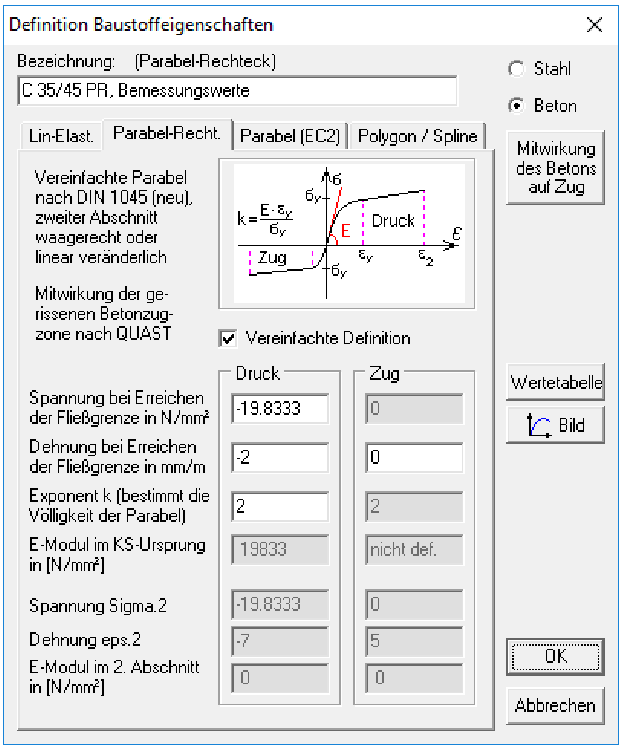
- Non-linear calculations, especially for cross-section checks (load-bearing capacity in ULS) with the stress-strain-functions according to EC2, for concrete and for steel / reinforcement
- Deformation calculations (moment-curvature) with correspondingly adjusted input values

Advantage:

- Simple definition of the usual stress-strain functions
- Adjustment of the shape of the parabola via the exponent k
- Adjustment of the slope in the second linear section (e.g. for the hardening area of reinforcing steel)

Disadvantage:

- Adaptation of the stress strain line to measured values from experiment is not always possible exactly



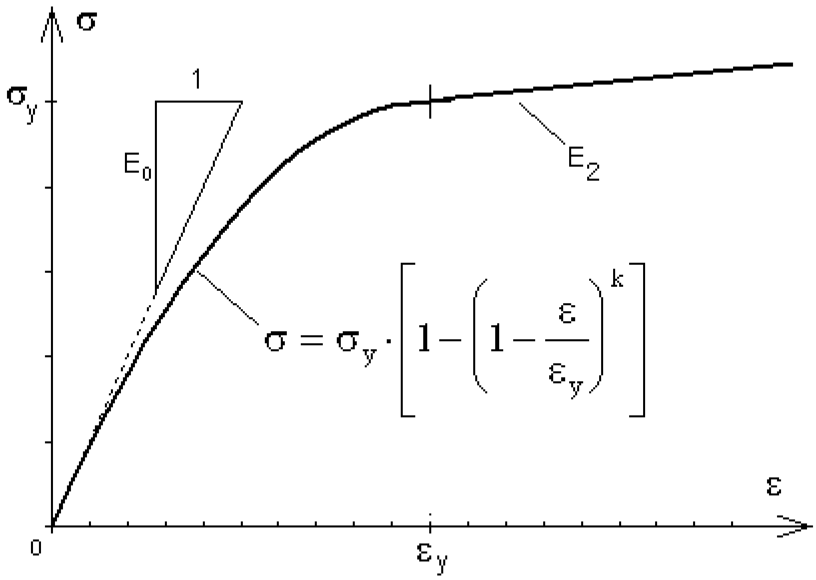
With this material law, you have the option of specifying separate branches for the compression side and for the tension side. However, the modulus of elasticity, which is recalculated when the input values are changed, should be the same at the coordinate origin for tension and for pressure.

The ***Simplified Definition*** check button makes it easier for the user if the second section of the material law is exactly horizontal and the modulus of elasticity in the coordinate origin has the same value on the tension and compression side. As you can see in the screenshot above, the input values for the second section will be grey. In addition, for the tension side, only the tensile strain ε_y needs to be entered, the corresponding tension is calculated by the program.

In the upper right corner, select the type of material (concrete or steel). In the case of concrete with tensile strength, please enter further parameters for the participation of the concrete in the cracked concrete tensile zone (button ***Participation of the concrete on tension***) for calculation of deformations with Stab2D-NL.

The advantage of this material definition lies in the simple adaptability of the parabola, which can be varied in wide ranges by the parameter k (exponent).

The usual parabolic rectangle diagram has been extended in the INCA2 program by making the second part linear rather than constant as usual. This makes it possible, for example, to easily adapt to the characteristic curve of the reinforcing steel, which rises slightly in the plastic range.



The relationship between strain and stress in the parabolic part is described by the following equation:

$$\sigma = \sigma_y \cdot \left[1 - \left(1 - \frac{\varepsilon}{\varepsilon_y}\right)^k\right]$$

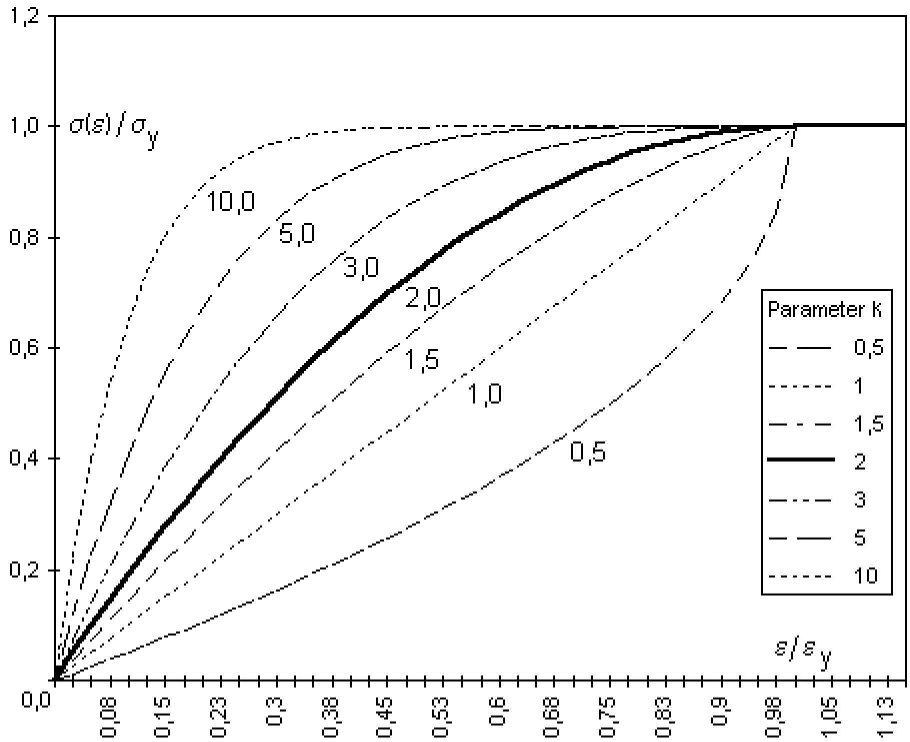
with $\sigma\psi$ = yield stress

ε_y = Yield elongation

Example of the parabolic rectangle diagram for different parameters k

with k = 0,5 1,0 1,5 2,0 3,0 5,0 10,0

(tension and strain normalized to 1.0)



By deriving the above-mentioned formula according to Epsilon, one obtains the modulus of elasticity as the slope of the function:

$$(E \cdot \varepsilon)' = (\sigma)'$$

$$E = \sigma_y \cdot k \cdot \left(1 - \frac{\varepsilon}{\varepsilon_y}\right)^{k-1} \cdot \frac{1}{\varepsilon_y}$$

The initial slope at $\varepsilon = 0$ is thus given as

$$E_0 = \frac{\sigma_y}{\varepsilon_y} \cdot k$$

When converted this formula, you also get the parameter k to a given E_0.

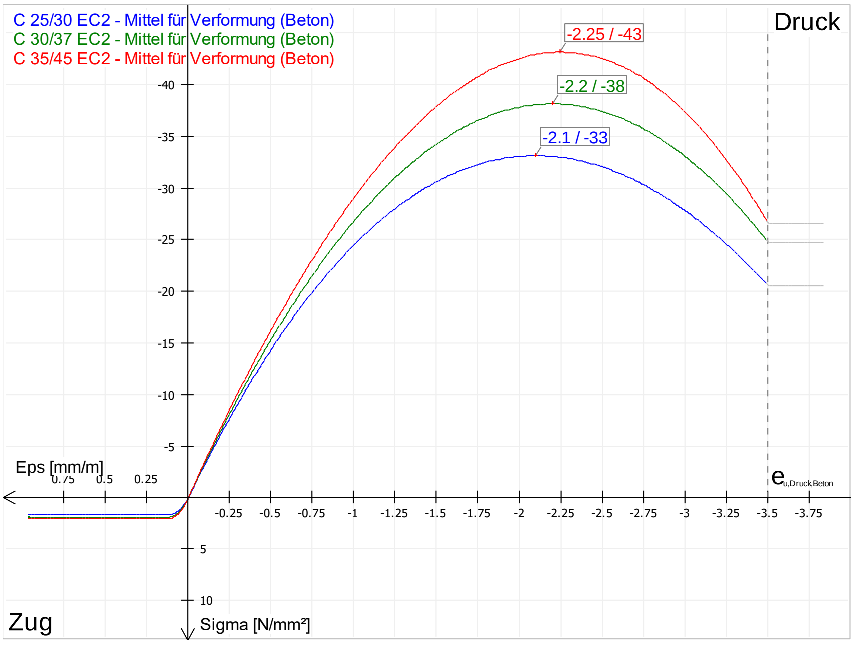
Please also read issue 415 DAfStb, pages 11 – 21.

Important for deformation calculations of reinforced concrete cross-sections (e.g. with the program Stab2D-NL) is the stiffening effect of the concrete in the cracked concrete tensile zone. Please read this point carefully as well.

6.3 Parabola (EC2)

Description:

- Pressure zone: Parabola-like, fractional-rational function according to EC2
- Tensile Area: Parabolic Rectangle Diagram (with Horizontal Second Section)
- Tension stiffening effect of concrete in the cracked concrete tensile zone according to the model of Quast and Espion



Application:

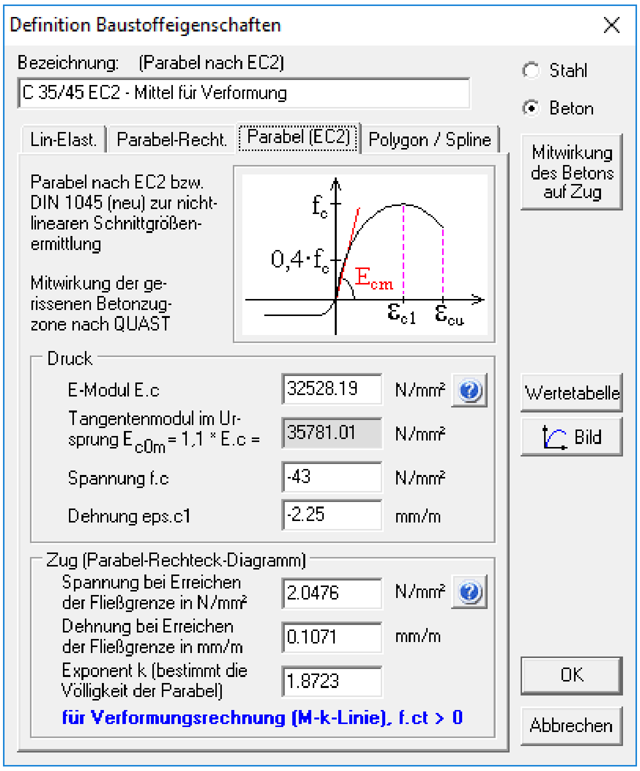
- **Nonlinear deformation calculations** as well as nonlinear / plastic force determination, e.g. with Stab2D-NL, calculation for the concrete with the stiffening effect of the cracked concrete tensile zone by applying a concrete tensile stress
- **Stress determination, especially in the state of serviceability,** e.g. for stress verification of concrete, verification of crack width, fatigue or similar.
In this case, the tension stiffening effect of the concrete is to be set to zero (f_{ct} = 0), so that the stress is determined in the crack!

Advantage:

- Simple definition of the stress-strain line for deformation calculations

Disadvantage:

- Pay attention to the definition range (limit strains!), because the function has another zero and an infinity point (asymptote). If the limit strain for concrete is defined too high on the pressure side, it can happen that a positive stress is determined despite negative strain. However, this circumstance is checked in INCA2 before each calculation and, if necessary, pointed out.
- Adjustment of the stress strain line (measured values from experiment) is not always possible exactly



The three parameters for the compression side can be found in the EC2. When entering the modulus of elasticity, it should be noted that the internal modeling in INCA2 is currently still done with a factor of 1.1 for the fractional-rational function. However, the current edition of the EC2 takes into account a factor of 1.05. Accordingly, the modulus of elasticity must be adapted, which has already been done in the predefined materials. See also the Excel file enclosed with the program.

The tension side for modelling the tension stiffening effect of the concrete tensile zone according to QUAST is defined as a parabolic rectangle diagram with a horizontal second branch.

In the upper right corner, select the type of material (concrete or steel). In the case of concrete with tensile strength, please enter further parameters for the participation of the concrete in the cracked concrete tensile zone (button ***Participation of the concrete on tension***). Please pay attention to the explanations there!

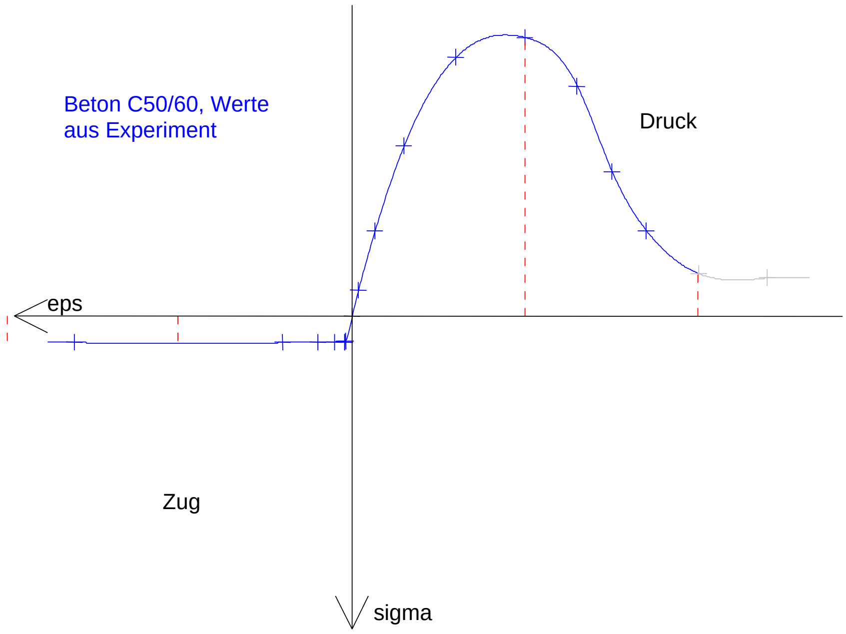
In the window area of the compression part, you will find an ***info button*** that provides you with further information about the modelling of the material. The correction of printing errors in DIN 1045-1 and the transition to the current EC2 are also briefly explained here.

With the second ***info button,*** you will receive brief information on the stiffening effect of the concrete on tensile tension and indications of useful input values. In blue, a guide is displayed further down the window. If a tensile strength is defined ($f_{ct} > 0$), then this material should preferably be used for deformation calculations (e.g. via M-k-lines or with Stab2D-NL). If no tensile strength is defined ($f_{ct} = 0$), then the calculation is carried out directly in the crack, so that the maximum stress and strain can be determined exactly, e.g. for verification of fatigue strength or crack width.

6.4 Polygon / Spline

Description:

- Polygon / Spline with arbitrary measured results from an experiment
- Tension stiffening effect in the cracked concrete tensile zone according to Quast and Espion



Application:

- Recalculation of experiments (deformation, load capacity) with exact material behaviour

Advantages:

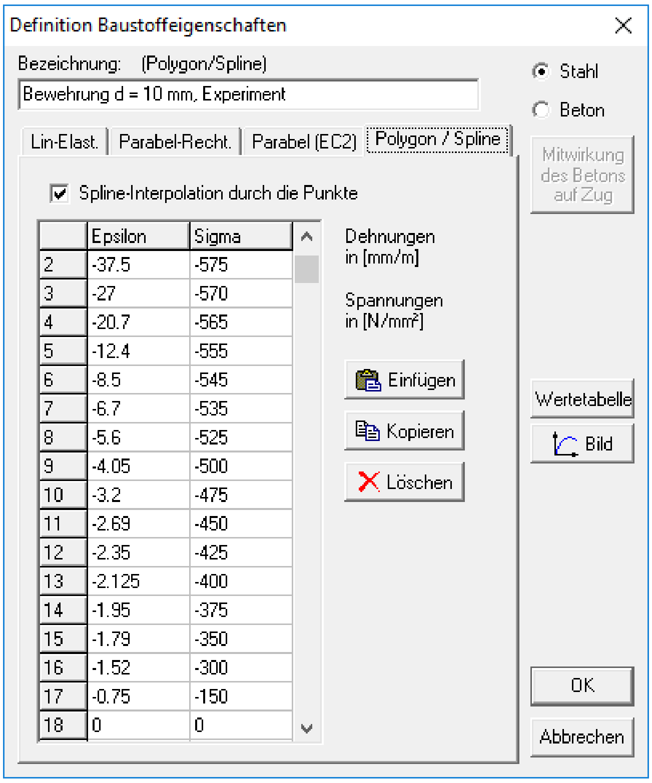
- Exact adaptation of the stress strain line to experimental data

Disadvantages:

- Time-consuming input due to many measured values (max. 100)
- with a large number of measured values, the calculation is slightly slower

In this window, you have the option to enter a maximum of 100 pairs of values (ε / σ). The order of the pairs of values when entering them does not matter, as they are ordered automatically.

The pairs of values describe a polygon through which a ***spline*** can be placed. The spline interpolation is done as usual with the boundary conditions that the curvature (2nd derivative) at the ends on the right and left is zero.



The "***Paste***" and "***Copy***" buttons can be used to paste values from Excel or copy them to the Windows clipboard. For example, it is very easy to determine the stress-strain line in Excel for hot design in steel construction or reinforced concrete construction depending on the temperature and copy it to INCA2 with a few mouse clicks.

6.5 Concrete – material characteristics

Since mistakes are often made in the modelling of concrete properties, here are a few words explaining how the definition according to Eurocode 2 (EC2) works.

6.5.1 Design Values for the Ultimate Limit State

These material parameters are used for the design of a reinforced concrete cross-section in the ultimate limit state (max. load-bearing capacity of cross-section). This covers the lower limit of the strength of the concrete, which can at least be expected even under the worst conditions.

To determine the strength according to EC2, the characteristic strength f_{ck} (5% fractiles) is divided by the partial safety coefficient γ_c= 1.5 and multiplied by an additional coefficient α_{cc} = 0.85 (α_{cc} according to the National Annex, country-specific, for Germany α_{cc} = 0.85).

For precast plants with factory production and special monitoring of concrete production, a reduced partial safety coefficient of γ_c = 1.35 may be taken into account.

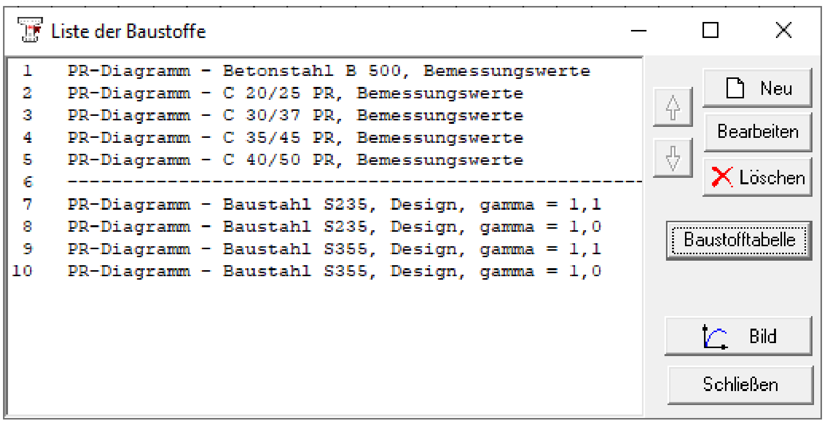
The tensile strength of the concrete is not taken into account in the design values, as the verification is carried out in the weakest cross-sectional part, i.e. in the crack, where only the concrete pressure zone and the reinforcement act to carry the load.

Example: The calculated value of the concrete strength for a C 30/37 is as follows:

$$f_{cd} = \frac{f_{ck}}{\gamma_c} \cdot \alpha = \frac{30 \frac{N}{mm^2}}{1{,}5} \cdot 0{,}85 = 17{,}0 \frac{N}{mm^2}$$

Since only the maximum stress/strength of concrete is reduced by a partial safety coefficient, but the associated elongation remains the same, the calculated modulus of elasticity of the concrete must consequently also be reduced. In the case of the aforementioned C30/37, the modulus of elasticity for the design value therefore has the value E_c = 17,000 N/mm².

In INCA2 and Stab2D-NL, the design values for concrete grades C12/15 to C100/115 are predefined and saved in the ***materials.inc*** file. Materials from this file can be transferred to the current file via the ***Materials Table*** button.



Possible adjustment of the partial safety coefficients for concrete:

1. Prefabricated parts

For pre-fabricated concrete parts with constant factory monitoring (parts with too little strength must be sorted out!).

$\gamma_c = 1.35$

2. Exceptional stress

In the event of an exceptional load situation, a smaller partial safety coefficient can be used for normal reinforced concrete:

$\gamma_c = 1.3$

3. Change of γ_c with increasing concrete strength (old DIN1045-1) – no longer valid!

In the older version of DIN 1045-1, edition 2001 (and subsequent editions, valid until 2012), an increasing partial safety coefficient was defined for concretes above C 50/60.

$$\gamma_c = \frac{1{,}5}{1{,}1 - \frac{f_{ck}}{500}} \geq 1{,}5$$

Example:	C55/67	$\gamma_c = 1{,}515$
	C100/115	$\gamma_c = 1{,}667$

4. Unreinforced components (DIN 1045-1)

$\gamma_c = 1.8$	For continuous/variable loads
$\gamma_c = 1.55$	For exceptional loads

5. Special structures for hydraulic engineering

According to DIN 19702 "Stability of solid structures in hydraulic engineering", depending on the design situation (BS-P / BS-T / BS-A), adapted partial safety coefficients may be used for the concrete and for the reinforcement.

All of the above changes (e.g. γ_c for pre-fabricated parts, adoptions according to old or other standards) must be carried out by the user himself. In particular, despite the European harmonisation of standardisation, deviations may occur, as other coefficients may be defined in the National Annex (e.g. α_{cc}). Even in the case of future adjustments to the standardizations, the material characteristics must be compared with the current status of the standardization and, if necessary, adjusted by the user. The materials listed in the ***Baustoffe.inc*** file should therefore always be checked for up-to-date and correctness!

6.5.2 Mean / Average values of material properties

The mean values of the material properties are used for the following **applications** :

1. Deformation calculations and internal force determinations of the overall system, taking into account the tension stiffening effect of the cracked concrete tensile zones, e.g. with the program Stab2D-NL for 2D-frame systems
2. Determination of stresses, in cracks with f_{ct} = 0,
   e.g. for fatigue calculations, crack width verifications, verification of stresses in the service condition

Case 1 – Deformation Calculations and Force Determination

For realistic deformation calculations and force determination of reinforced concrete structures, the determination of a realistic, stress-dependent stiffness is of great importance. For this purpose, the average / mean values of the materials are used, which represent the average material behaviour.

Reinforced concrete beams, which are normally stressed, are cracked (condition II), so that realistic modelling of the cracked concrete tensile zone is required. The so called tension stiffening effect is resulting from the concrete between two cracks where it is still uncracked. The concrete is in condition I and thus the beam has a higher stiffness at this point than in the crack next to it. When calculating with concrete tensile strength, the method of QUAST and ESPION of "smeared cracks", in which an average concrete tensile stress is applied and thus the cracks and the intervening uncracked areas are "averaged". Good empirical values for this smeared tensile stress are 1/20 to 1/30 of the compressive stress and 1/20 to 1/30 of the associated elongation. Further explanations of the Tension stiffening effect can be found under point ***6.7 Interaction of concrete in the cracked concrete tensile zone***.

The material file delivered with INCA2 contains the average values of all common types of concrete according to EC2. The tensile stress $f_{ct,cal}$ or the tensile strain $\varepsilon_{ct,cal}$ are useful empirical values of the authors, but they can fluctuate quite strongly in individual cases. Poor post-treatment, early drying of the concrete, frost effect on the young concrete or similar can significantly reduce the stiffening contribution of the concrete tensile zone. The model for the reduction of tension for higher strains (decrease in the case of greater strains) should also be adjusted by the user if necessary.

Case 1a – General Deformation Calculations and Internal Force Determination

This case includes all beam systems or moderately slender frame structures that do not have a significant influence from 2nd order theory. In this case, the mean values of the material properties predefined in the ***materials.inc*** file should be used.

Case 1b – Column Design (2nd Order Theory)

When determining the internal forces for a column or slender frame systems, the result depends very much on the deformation of the system in the case of great slenderness. This is a problem that is influenced by deformation. A poor concrete with a low modulus of elasticity causes a large deformation and thus an increase in stress. Since columns are relatively small components, it can easily happen that the entire component is made from a margin of bad concrete. In this case, since the strength and stiffness are at the lower limit of the usual range and these values represent a safety-relevant input variable, the values must be reduced by γ_c. The strength of the concrete for the deformation calculation is then to be used as follows:

Strength $f_c = f_{Cm} / \gamma_c$

Modulus of elasticity $E_c = E_{Cm} / \gamma_{CE}$

with f_{cm} = average value of concrete compressive strength

$\gamma_c = \gamma_{CE} = 1.5$ for Germany, see National Annex to EC2

Example: The calculation value of the concrete compressive strength for a C 30/37 is thus given by:

$$\frac{f_{cm}}{\gamma_c} = \frac{38\,\frac{N}{mm^2}}{1{,}5} = 25{,}33\,\frac{N}{mm^2}$$

with $f_{cm} = 30 + 8 = 38$ N/mm²

Tensile strength is taken into account, as the tension stiffening effect of the concrete between the cracks is still present even in the case of "bad" concrete. Here, too, the empirical value of 1/20 to 1/30 of the compressive strength values can be assumed.

In such a calculation, however, it is important that the rated resistance of the cross-section is not exceeded. Since the M-k-line was determined with a computational strength $f_{cd,\,column} = f_{cm} / 1.5$ (with f_{cm} and without coefficient $\alpha_{cc} = 0.85$), the maximum bending moment of the Mk line will be above the bearing capacity (bending moment M_{Rd}). For this reason, after calculating the deformation and internal forces, a safety check with $f_{cd} = \alpha_{cc} \cdot f_{ck} / 1.5$ must be carried out so that the cross-section can also carry the determined loads.

As you can see, you are dealing with 2 different material properties here, between which you have to switch back and forth depending on the calculation process. A corresponding example with possible simplifications is also included in the examples in the following chapters.

See also EC2-1-1, section 5.8.6:

5.8 Berechnung von Bauteilen unter Normalkraft nach Theorie II. Ordnung

. . .

5.8.6 Allgemeines Verfahren

(1)P Das allgemeine Verfahren basiert auf einer nichtlinearen Schnittgrößenermittlung, die die geometrische Nichtlinearität nach Theorie II. Ordnung beinhaltet. Es gelten die allgemeinen Regeln für nichtlineare Verfahren nach 5.7.

(2)P Für die Schnittgrößenermittlung müssen geeignete Spannungs-Dehnungs-Linien für Beton und Stahl verwendet werden. Kriechauswirkungen sind zu berücksichtigen.

(3) AC Die in 3.1.5, Gleichung (3.14) und 3.2.7 (Bild 3.8) AC dargestellten Spannungs-Dehnungs-Linien für Beton und Stahl dürfen verwendet werden. Mit auf Grundlage von Bemessungswerten ermittelten Spannungs-Dehnungs-Diagrammen darf der Bemessungswert der Tragfähigkeit direkt ermittelt werden. In Gleichung (3.14) und im k-Wert werden dabei f_{cm} durch den Bemessungswert der Betondruckfestigkeit f_{cd} und E_{cm} durch

AC $E_{cd} = E_{cm} / \gamma_{CE}$ AC (5.20)

ersetzt.

ANMERKUNG Der landesspezifische Wert γ_{CE} darf einem Nationalen Anhang entnommen werden. Der empfohlene Wert ist 1,2.

NDP zu 5.8.6 (3)

Dabei ist $\gamma_{CE} = 1,5$

Die Formänderungen dürfen auf der Grundlage von Bemessungswerten, die auf den Mittelwerten der Baustoffkennwerte beruhen (z. B. f_{cm}/γ_C, E_{cm}/γ_{CE}) ermittelt werden. Für die Ermittlung der Grenztragfähigkeit im kritischen Querschnitt sind jedoch die Bemessungswerte der Baustofffestigkeiten anzusetzen.

Für die Aussteifungskriterien nach 5.8.3.3 gilt $\gamma_{CE} = 1,2$.

(4) Fehlen genauere Berechnungsmodelle, darf das Kriechen berücksichtigt werden, indem alle Dehnungswerte des Betons in der Spannungs-Dehnungs-Linie gemäß 5.8.6 (3) mit einem Faktor $(1 + \varphi_{ef})$ multipliziert werden. Dabei ist φ_{ef} die effektive Kriechzahl gemäß 5.8.4.

(5) Die günstigen Auswirkungen der Mitwirkung des Betons auf Zug dürfen berücksichtigt werden.

ANMERKUNG Diese Auswirkung ist günstig und darf zur Vereinfachung immer vernachlässigt werden.

NCI zu 5.8.6 (5)

ANMERKUNG Diese Auswirkung ist nur bei Einzeldruckgliedern immer günstig.

(6) Üblicherweise werden die Gleichgewichtsbedingungen und die Dehnungsverträglichkeit von mehreren Querschnitten erfüllt. Werden vereinfachend nur die kritischen Querschnitte untersucht, darf ein realistischer Verlauf der dazwischen liegenden Krümmungen angenommen werden (d. h. ähnlich dem Momentenverlauf nach Theorie I. Ordnung oder entsprechend einer anderen zweckmäßigen Vereinfachung).

Modulus of elasticity:

According to EC2, the modulus of elasticity depends only on the strength class of the concrete. However, due to different surcharges depending on the region, considerable fluctuations can occur. In general, the values in German standardization tended to be at the upper limit of the fluctuation range. At this point, reference should be made to an article in the journal Beton [6/2003, pp. 294 - 298], "Proficiency test for the determination of the static modulus of elasticity of concrete" by Brameshuber and Brockmann, Aachen.

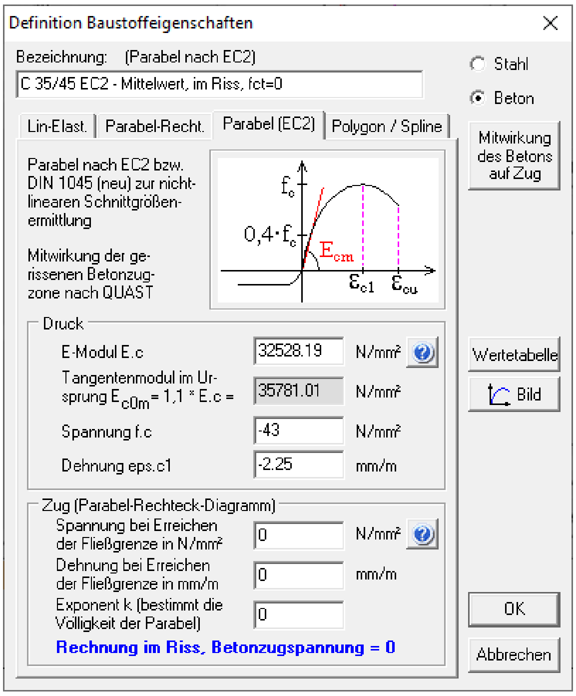
The regulations in Switzerland are exemplary in this respect, where different values have been set depending on the region. In extreme cases, the values for the modulus of elasticity can therefore fluctuate by 50% !!

In German standardization, the modulus of elasticity has been adapted again and again over several generations of standards. With regard to the aggregates available in northern Germany, the current standard values for the modulus of elasticity are, in the opinion of the author, somewhat too high.

If a high level of computational accuracy is required, the values for the modulus of elasticity should therefore be determined experimentally or should be demanded from the concrete supplier. The correct assumption of the material parameters is particularly important when determining internal forces in the case of problems affected by deformation (e.g. slender valley supports in bridges).

Case 2 - Determination of stresses, in crack with f_{ct} = 0

For case 2, the calculation in INCA2 takes place directly in the crack. For this purpose, the predefined average values can be used, but the concrete tensile strength must be set to zero manually. It is used, for example, for fatigue calculations, crack width checks or verification of stresses in the service state.



6.6 Application of materials in Stab2D-NL and INCA2

In combination with the programs Stab2D-NL and INCA2, some standard cases can be defined in which the material characteristics are to be applied as follows:

1. Deformation calculation and internal force determination for normal systems

- Modelling in Stab2D-NL:
  - Mean values for concrete, incl. tension stiffening effect of the concrete
  - Mean values for reinforcement
- Checking of the cross-sectional load-bearing capacity with INCA2: After determining the forces in Stab2D-NL, the cross-sections with the design values (concrete + reinforcement) are to be modeled separately with INCA2 and the internal forces determined in Stab2D-NL are to be applied. This is used to check the cross-sectional load capacity.

2. Deformation calculation and internal force determination on slender systems (columns)

e.g. columns at risk of buckling, load-bearing members with compressive force and effects from Th. 2nd order

2.a Columns with accurate modelling (so called "double book-keeping")

- Modelling in Stab2D-NL:
  - Column values for concrete, incl. tension stiffening effect of the concrete
  - Design values for reinforcement
- Check of the cross-sectional load capacity with the design values in INCA2, as in 1.

This method is often referred to as "double book-keeping" because 2 different material parameters are used for the calculation. This method generally provides the highest load capacities for slender systems.

Under certain conditions, it is possible to set the concrete characteristics higher for the deformation calculation, e.g. in the production of columns in the precast factory. However, this requires a very close check of the concrete compressive strengths and the concrete modulus of elasticity, as well as appropriate coordination with the supervisory authority.

2.b Columns with simplified modelling

- Modelling in Stab2D-NL:- Design values for concrete- Design values for reinforcement
- There is no need to check the cross-sectional load capacity in INCA2, as the design values of the material properties have already been calculated.

This method is very conservative and therefore provides the smallest load-bearing capacities for the column. However, the procedure is very simple and not prone to errors.

2.c Simplified Modelling, Design Values with Tension Stiffening

- Modelling in Stab2D-NL:
  - Design values for concrete, but supplemented by tension stiffening effect of the concrete
  - Design values for reinforcement
- The check of the cross-sectional load-bearing capacity in INCA2 can usually be omitted, since the design values have already been calculated and in the limit state of the cross-sectional load-bearing capacity when the yield point of the reinforcement is reached, the stiffening effect of the concrete is set to zero.

This method provides slightly smaller load capacities than variant 2.a and slightly larger than variant 2.b. However, the user has to model the tension stiffening effect of the concrete to tension in the concrete by himself. It is usually sufficient to define a tensile elongation for concrete of ε = 0.1 mm/m.

This method is comparatively simple and fast to model, but requires safe handling of tensile stiffening in concrete.

3. Deformation calculation and internal force determination on systems with plastic hinges

The typical system for this is the 2-span beam, in which the cross-sectional load capacity above the central support is reached. With increasing load, a plastic joint is formed there and the load can then be further increased until the cross-sectional load capacity is also reached in the field.

Regardless of the material characteristics explained below, all aspects influencing the rearrangement should be taken into account in the modeling in Stab2D-NL, including:

- Modelling of the width of the central support to “round” the moment
- Load distribution for a single load up to the axis of the beam, application of the load as a line load instead as a single load
- Modelling reinforcement: In addition, define the rise in the plastic area, so that even after the yield point has been exceeded, the stress can still rise slightly. As a result, the stiffness is slightly greater, so that the calculation with Stab2D-NL is much more stable.

3.a Simplified modelling system with design values such as 2.b or 2.c

A simple modelling to obtain the effect of internal moment redistribution on statically indeterminate systems is modelling with design values (concrete + reinforcement) in Stab2D-NL, supplemented in concrete by the stiffening contribution of the cracked concrete tensile zone. For the calculation, only the modeling and calculation in Stab2D-NL is required. A separate proof in INCA2 is not required.

In this modeling variant, the deformation of the system is determined to be significantly too large, since the "poor" design values are used for the entire system. This modelling is therefore not suitable for determining the real deformation and realistic behaviour for redistribution of bending moments.

3.b System with adjusted mean values

The basic idea for this variant is to depict the system as realistically as possible and therefore with the mean values, so that realistic deformations can be determined in the service state.

If the cross-sectional load capacity is reached at the maximum stressed point, e.g. at the centre support area of a two-span girder, the system should be softer at exactly this point in order to enable redistribution of bending moment. However, the moment at the support in the centre should not be greater than the cross-sectional load capacity with the design values. In order to achieve this, the system would actually have to be modelled with the design values at the relevant point, but this contradicts the realistic modelling with the mean values.

In order to circumvent this contradiction, the materials are modeled and adapted in Stab2D-NL as follows:

- Concrete modelling with the mean values
  - adjustment of the compression concrete stress f_c from f_{cm} to the design value f_{cd}
  - modulus of elasticity and stiffening contribution of the concrete to tension remain unchanged
- Reinforcement- Reinforcement with design values, but additionally define the rise in the plastic area

With this modelling, it is achieved that the mean values of the material properties are used for loads below the cross-sectional load capacity. If the cross-sectional load-bearing capacity is reached, the reinforcement starts to yield when f_{yd} is exceeded, the cross-section becomes softer and thus allows relocation to other areas of the system.

Subsequently, a check of the cross-sectional load capacity with INCA2 (as in 1.) must be carried out. In this case, the design values are to be used, in which the plastic branch of the reinforcement is also to be modelled with a rising slope. The cross-sectional load capacity is usually slightly exceeded in the highly stressed areas of this design, but this can be accepted.

In the case of concrete, it can be advantageous to approximate the stress-strain line using the parabolic rectangular law instead of the parabola-like function from EC2 for internal force determination. The advantage is that after reaching the maximum stress, the σ-ε line of concrete does not drop down and the calculation is therefore more stable.

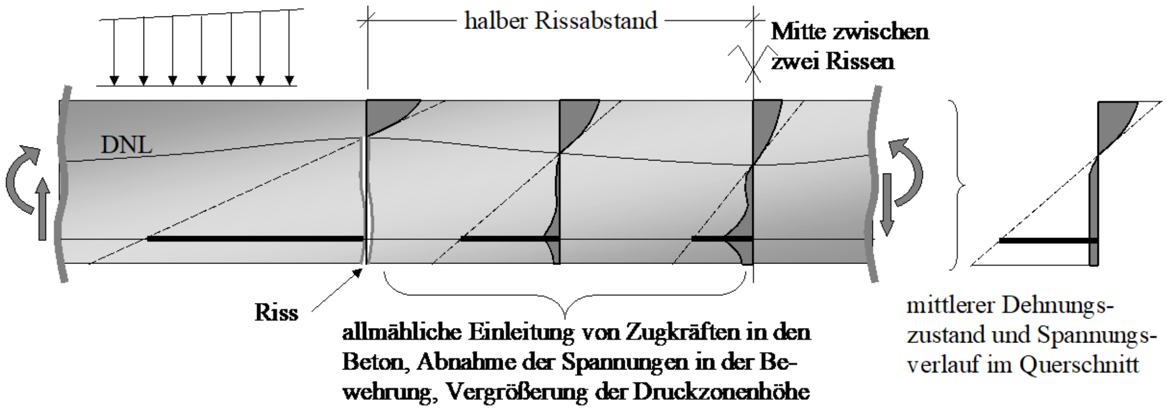
3. Recalculation of experiments

For this purpose, the values of the materials measured in the laboratory must be entered. If these are only partially known, only the mean values should be used for both the deformation calculation and the cross-sectional load-bearing capacity. Deviations of the values should be taken into account, so that upper and lower values should be expected. This applies in particular to the tension stiffening effect of the concrete tensile zone, which experience has shown can fluctuate greatly.

6.7 Contribution of concrete to tension in the cracked concrete tensile zone

6.7.1 Stress Distribution in the Cracked Tensile Zone

The stress distribution in the cracked concrete tensile zone can be approximated according to the following graphic. Precisely within the crack the entire tensile force is carried by the reinforcement, the compressive force by the concrete. Next to the crack there is an uncracked area, which receives tensile stresses again from the reinforcement via the composite stress between the concrete and reinforcement. The resulting stress is concentrated in the immediate vicinity around the reinforcement. The further you move away from the crack, the higher the concrete tensile stress will be. As the stress increases, the tensile stress in the concrete will increase until the concrete tensile strength is reached and a new crack forms at this point.



The resulting stress state on the tensile side, which varies greatly in detail, is simplified to a roughly constant concrete stress for the beam calculation, as shown on the right-hand side in the graphic. Depending on the stress on the cross-section, the real stress distribution in the tensile zone changes, so that the simplified constant stress must also be adjusted.

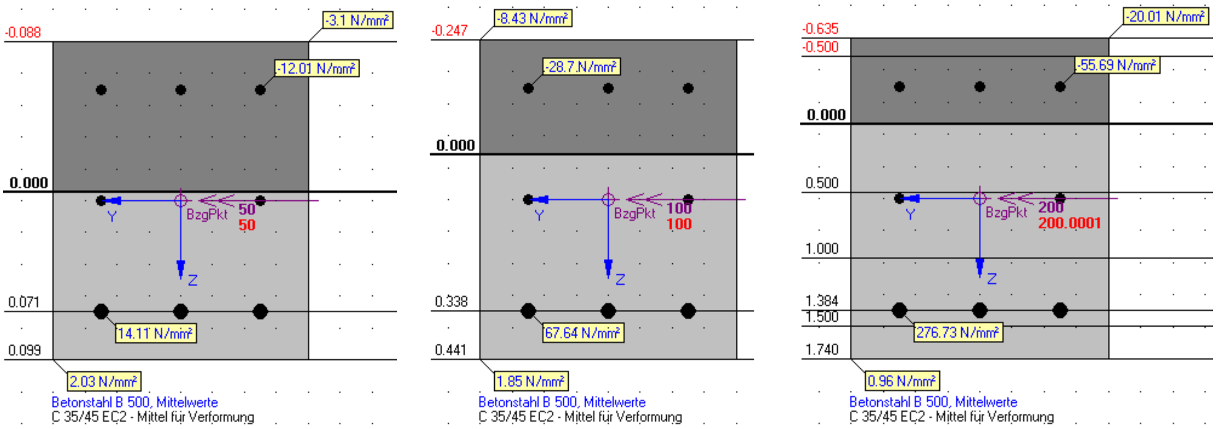
6.7.2 Modeling in INCA2

Since the calculation of single, discrete cracks in beams only increases the computational effort, but not noticeably the accuracy, the method of "smeared cracks" is used in the INCA2 program and Stab2D-NL. In this process, a simplified stress is applied in the concrete tensile zone, which is adjusted depending on the elongation of the tensile reinforcement. This makes it possible to determine deformations of reinforced concrete beams or slabs with acceptable accuracy and with a reasonable computational effort.

This method was proposed by QUAST (1977/80/81), among others, and extended by ESPION (1985) to include a nonlinear progression. Recalculations of 39 beam tests showed a sufficiently good agreement for practical construction concerns. You can read more about this and other procedures in a publication by Quast/Busjaeger. An excerpt from issue 415 of the DAfStb is included at the end of this section. There you will also find some statements on the limits of application of the procedure.

Increasing load for a beam cross-section

M = 50 kNm M = 100 kNm M = 200 kNm

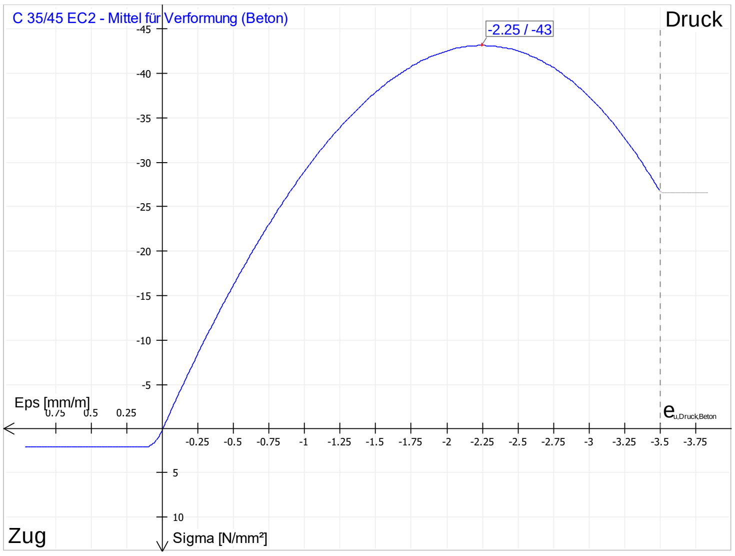


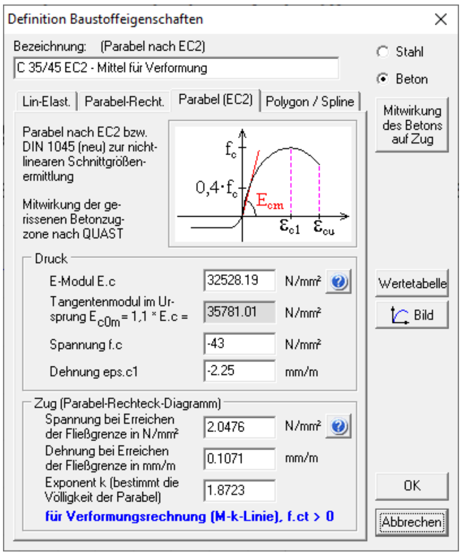
$FC_{\text{Train side}}$ = 2.03 N/mm² 1.85 N/mm² 0.96 N/mm²

roughly uncracked shortly after cracking serviceability

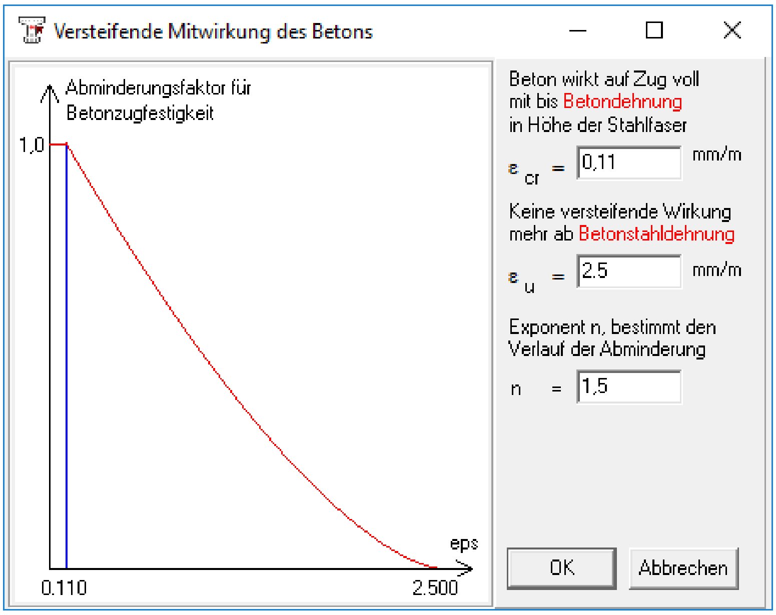
Calculation procedure

On the tensile side, a chosen tensile stress $f_{ct,cal}$ is defined as a parabolic rectangle for the concrete. At the coordinate origin, the modulus of elasticity on the tension side and the modulus of elasticity on the compression side have the same value.





Depending on the cross-sectional load, the tensile stress in the concrete is reduced by a factor resulting from the following function:



On the x-axis, the concrete strain ε plotted. This concrete strain is determined in the calculation at the point of the steel fibre with the max. tension.

Condition I: Up to strain ε_{Cr} (crack = cracking, exceeding the tensile strength) the concrete is fully acting (completely uncracked).

Condition II: If this strain ε_{Cr} is exceeded, the tensile stress in the entire tensile zone is reduced by the determined factor, i.e. the parabola-rectangular diagram on the tension side is reduced in the direction of the σ –axle. Also, the area near the strain zero line with strains less than ε_{Cr} is reduced in size with this factor.

Condition III: If the yielding point of the steel is exceeded, the concrete generally no longer contributes or only slightly. A good value for $\varepsilon\upsilon$ is therefore $\varepsilon\upsilon$= 2.5 mm/m. If the concrete elongation at the reference point is greater than the value of $\varepsilon\upsilon$, the tensile stress in the concrete is completely set to zero.

Empirical values for the stiffening contribution of the concrete tensile zone $f_{ct,cal}$

The relevant input value is the smeared value of the tensile stress $f_{ct,cal}$. This value was chosen according to recalculations of several deformation-tests in such a way that the best possible agreement was obtained for common beam and column systems. Due to the chosen mathematical modelling of the tensile stiffening effect according to Quast and Espion, the value $f_{ct,cal}$ does not correspond to the usual concrete tensile strength f_{ctm}, but is approximately $f_{ct,cal}$ = 60% · $f_{ct,medium,EC2}$.

Definition of tensile behaviour of concrete (material law)

$f_{ct,cal}$ = 1/20 · f_{cm} to 1/30 · f_{cm} similar values are provided by the approach $f_{ct,cal}$ = 60% · $f_{ct,medium,EC2}$	Tensile strength, empirical value
ε_{cr} = 1/20 · ε_{cu} to 1/30 · ε_{cu}	Elongation when tensile strength is reached, empirical value
n = 1.5 . . . 2,0	Exponential for the shape of the parabola, should be chosen in such a way that approximately the same modulus of elasticity in the coordinate origin results on the tensile and on the compression side.

Definition of the factor

ε_{cr} = 1/20 · ε_{cu} to 1/30 · ε_{cu}	Strain when tensile strength of concrete is reached (as previously with the concrete material law)
$\varepsilon\upsilon = \varepsilon_{s,y}$	The choice of $\varepsilon\upsilon$ as the yielding point with e.g. $\varepsilon_{s,y}$ = 2.5 mm/m for a B500 is a computational simplification. The real contribution will have fallen to about zero at an elongation between 2.0 and 3.0 mm/m. Therefore, for simplification, $\varepsilon\upsilon = \varepsilon_{s,y}$. However, in the case of higher-strength reinforcement, this value should not exceed 3 mm/m or only if the exponent n is correspondingly increased to adopt the shape.
n = 1.0 . . . 3,0	Exponent for the shape of the reduction curve, by default n = 1.0 is given. However, even a small amount of previous damage (temperature stress, previous load) is sufficient to achieve better results with n = 2.0. If the n is larger, the participation of the tension zone is reduced more quickly.

6.7.3 Limits of application

The method was developed for the calculation of normally reinforced concrete cross-sections under bending with and without axial force. Pure tension rods are somewhat harder to model with the standard values and require a certain amount of adaptation in terms of the material parameters.

Cross-sections with very small reinforcement or cross-sections without tensile reinforcement cannot be calculated according to this model, or the results are incorrect. In this case, an adjustment should be made to the cross-section itself by reducing the size of the effective concrete tensile zone around the reinforcing bars accordingly.

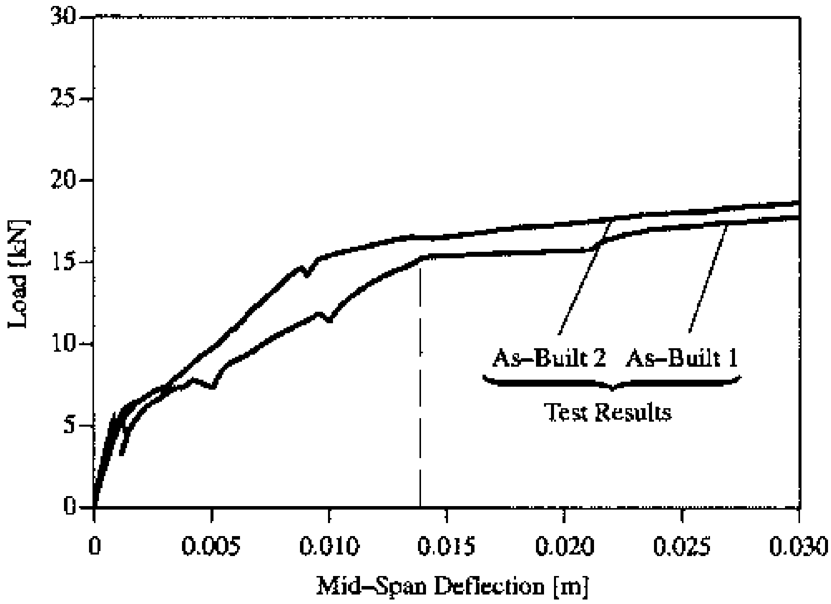
Furthermore, by using $f_{ct,cal}$ = 1/20 * f_c to 1/30 * f_c, the crack moment $M_{crack\ is}$ slightly underestimated, the cracks open a little too early. This means that the deformation of beams, which are subjected to approximately M_{crack}, is determined to be somewhat too large. However, since the calculated tensile stresses in the cross-section are only slowly reduced with a further increase in load, the behaviour of the beam can still be determined quite well shortly after cracking.

In any case, it would be wrong to use the average tensile strength f_{ctm} taken from DIN or EC2 for $f_{ct,cal}$! Due to the special formulation of the tension stiffening according to Quast / Espion, the stiffness of the concrete cross-section would be significantly overestimated!

Further adjustments and tendencies

If two identical bars are tested, there are already differences in the load deformation diagrams, even though the bars are nominally identical. The recording of the actually existing material properties (conditions on the construction site!) is therefore rather a stroke of luck. In any case, it makes more sense to expect the most "good" and "bad" properties of materials. This gives us the range in which the deformations are to be expected.

Load-deformation curves for 2 "identical" plates with three-point bending test



Above all, modulus of elasticity and tensile strength can vary greatly and also result in the greatest fluctuations in the resulting deformations in the service state. Even temperature stresses (e.g. low temperatures shortly after casting) or early drying of the surface due to inadequate post-treatment lead to pre-damage and thus to a decrease in tensile strength and thus in the stiffening effect.

Furthermore, it should be noted that with an increase in concrete compressive strength, the tensile strength does not increase to the same extent, but rather disproportionately. In the case of higher-strength concretes, an estimation with e.g. $f_{ct,cal} = 1/25 \cdot f_{cm}$ or $f_{ct,cal} = 1/30 \cdot f_{cm}$ gives possibly better results. It should be noted here that most of the beam tests calculated by Quast showed normal to low concrete strengths – at least from today's point of view of usual concrete strengths.

The frequently performed expansion tests, which are used to determine the interaction of the cracked concrete tensile zone, provide a certain stiffening effect after yielding of reinforcement. However, calculations on bending beams show that modelling without tensile stiffening after yielding of reinforcement is a sufficiently good approximation.

In sum, it should be noted that with the model according to Quast and Espion used in INCA2, only calculations of beams and columns are useful. Tension rods made of concrete are therefore less easy to calculate, but they are also rather rare in practice.

In general, it can be said that the method of stiffening participation implemented in INCA2 is a sufficient approach for practical construction calculations, because on the one hand it is easy to use, provides acceptable values in comparison with experiments and provides "nice" moment-curvature lines without jagged edges and jumps for a more advanced deformation calculation.

6.7.4 Behaviour in case of pre-stretching of concrete or reinforcement (e.g. shrinkage)

Pre-stretched concrete: From the current cross-sectional elongation and the pre-expansion of the concrete, the material elongation of the concrete is determined for stress calculation. Does this concrete-material elongation exceed the elongation ε_{Cr} , the tensile stress is reduced by the appropriate factor. The factor becomes zero when the steel at this point exceeds the strain $\varepsilon\upsilon$.

Pre-stretched reinforcement: Here, too, as before, the concrete elongation is first calculated at the point of the determining reinforcement fibre. Until ε_{Cr} there is no reduction in concrete tensile stresses. Then follows the part of reduction according to the curve. If the steel-strain (= cross-sectional elongation + pre-elongation of the reinforcement) exceeds the value $\varepsilon\upsilon$, the reduction becomes zero.

This behaviour can be clearly demonstrated using the example of a rectangular cross-section with reinforcement at the bottom and shrinkage of the concrete with ε_{cr} = 0.4 mm/m. The shrinkage strain can be applied to the concrete on the one hand as tensile pre-strain, but on the other hand it can also be applied to the reinforcement as compressive pre-strain. In the end, both models provide the same result in the reduction of the tensile stresses of the concrete tensile zone and thus the same Mk lines for a deformation calculation.

The first option (tensile pre-expansion on the concrete) has the advantage that it is easy to handle in INCA2 and is also taken into account exactly where shrinkage occurs - namely in the concrete. However, the disadvantage is the display of the strain plane in the subsequent graphical output. The size of the "pressure zone" may be a little confusing. However, it must be borne in mind that the elongation plane is no longer represented in relation to the concrete, but in relation to the non-pre-stretched cross-sectional parts (in this case, reinforcement).

The second option is to apply the pre-extension to all reinforcing bars, but now as a compression pre-expansion with ε = -0.4 mm/m. The graphic representation looks a bit more familiar, as you can now really see how big the pressure zone and how big the tensile zone of the concrete is.

In the sample files

Schwinden_auf_Beton.inc

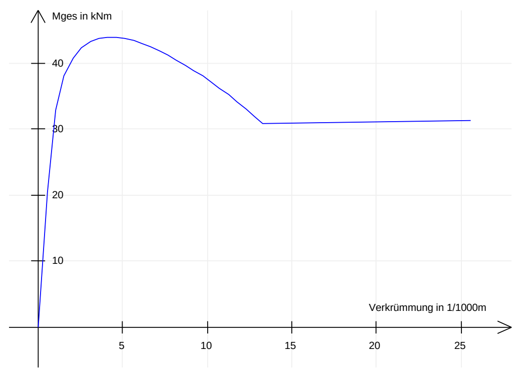
Schwinden_auf_Stahl.inc

these different modellings have been carried out. More detailed information can be found in the chapter *6.12 Shrinkage and creep*.

6.7.5 Application Issues and Solutions

Fundamentally, problems with the tension stiffening law for the concrete tensile zone will arise if the scope of the standard modeling is left behind. Looking back at a steady e-mail exchange with various users, the following focal points have emerged:

#	Application problem and impact	Solution
1.	For the calculation value of the tensile strength $f_{ct,cal,}$ the average value of the tensile strength of the concrete according to EC2 is used (f_{ct}). As a result, the stiffening contribution is overestimated.	Use of the predefined mean values of the material properties for the standard concretes. In the case of self-defined concretes, approx. 1/20 to 1/30 of the compressive strength is to be used as the calculation value of the tensile strength. Similar values are also provided by the approach with $f_{ct,cal}$ = 60% · f_{ct},Medium,EC2 .
2.	Low or under-reinforced cross-sections: The load-bearing capacity of the uncracked cross-section (M_{crack}) is higher than the load-bearing capacity of the cracked cross-section (M_{yield}). In this case, the Mk line has a "hump" as shown in the graph below.	In the case of a force-controlled test setup (e.g. 3-point bending test), the beam would fail completely immediately after the initial crack formation. However, since most of the tests are displacement-controlled, a load-deformation curve can still be determined. In INCA2 or Stab2D-NL it is possible to try to reduce the size of the concrete tensile zone so that the crack moment is calculated to be

		smaller. In addition, the calculated tensile stress $f_{ct,cal}$ could also be reduced. In such a case, the minimum reinforcement must be checked for everyday engineering (away from the laboratory) in order to avoid failure without prior notice.
3	Modelling of concrete with the mean values and reinforcement with the design values. In this case, the reinforcement starts to yield at an elongation of ε_s= 2.174 mm/m. However, the Participation Act specifies a flow elongation of ε_s = 2.5 mm/m. As with point [2], this results in a hump for the Mk line, so that it cannot be used for further calculations.	Uniform use of material parameters, i.e. for both concrete and reinforcement with mean values (or design values)

6.7.6 Excerpt from issue 415 of the DAfStb, Beuth Verlag, Berlin, 1990

'2.2.1.2 Concrete tensile zone

Calculations in the uncracked state I are approximately based on the linear theory of elasticity. The occurring concrete tensile stresses can be easily taken into account within the framework of a program-controlled calculation by specifying a linear stress expansion line until the flexural tensile strength of the concrete is reached. The shape of the stress strain relationship for the concrete tensile zone then corresponds to the representation in Figure 2.3 for the exponent n = 1. In this way, for example, a stress check of a cross-section under limited preload can be carried out in the service condition (see also Example 4 in Annex 1, Chapter 1.4.4).

The stiffening effect of the concrete on tension between the cracks in condition II is attempted with different methods. With regard to the expediency of these procedures, a distinction must be made between two fundamentally different areas of application:

- *theoretical studies in research and*
- *practical calculations.*

For research purposes, complex methods based on the finite element method are generally used. They provide insights into the basic mechanical relationships, but are of secondary importance for practical construction concerns due to the high effort involved in system mapping and calculation. For example, the bond behavior can be recorded more or less well in the context of a finite element calculation *by coupling steel and concrete elements by means of suitable composite elements. Micro-elementation can also be used to investigate the behaviour of discrete cross-sectional parts, whereby the restrictive requirements of the technical beam bending gauge (keeping the cross-sections flat) are overcome. An overview of finite element methods for calculating reinforced concrete structures is given by Eibl and Ivanyi (1976).*

. . .

In the context of some deflection calculations of simple reinforced concrete slabs using the finite element method, Gilbert and Warner (1978) investigate the usability of three different stress strain relationships for the concrete in the tensile zone. In addition, these calculations are also carried out with a modified stress strain line of the reinforcing steel, taking into account a modulus of elasticity that varies section-by-section depending on the strain at the crack. The authors suggest that in cases where the amount of reinforcement differs, the latter approximation method would probably result in a better agreement with the actual conditions.

Quast (1977/80/81) extends the usual parabolic rectangular diagram according to DIN 1045 (1988) to the general stress expansion line for concrete shown in Figure 2.4 and uses this line defined in the tensile and compression range to recalculate the moment-curvature relationships of a total of 39 beam tests, resulting in a sufficient agreement with the tests for practical purposes. Espion (1985) adopts this definition of the stress strain line for concrete, but takes into account a nonlinear decrease in the variable mean value of concrete tensile stresses according to equation (2.3) in a calculation program developed on the basis of the finite element method. The approach of a quadratic reduction of the mean value of the concrete tensile stresses often results in a better agreement with the test results for the tests it recalculates (pure tensile tests as well as beam, column and frame tests).

Schwennicke (1983) describes another way of determining the stiffening effect of the concrete in the tensile zone by defining a suitable stress expansion line of the concrete. Grzeschkowitz (1988) adopts this approach for the recalculation of tests on compression columns with rectangular cross-sections. The used σ-ε relationship of the concrete tensile zone shows a linear increase until the concrete flexural tensile strength is reached and a subsequent sloping, nonlinear course up to an elongation at which the concrete's contribution to tensile strength ends. In order to determine the exponent of the parabolic function for the sloping branch, the mean concrete flexural tensile strength, the composite stress, the modulus of origin of the concrete, the modulus of elasticity of the reinforcing steel, the degree of reinforcement of the concrete tensile zone, the bar diameter of the reinforcement and the limit elongation for the contribution of the concrete to tensile action are required. The test recalculations carried out by Grzeschkowitz show good agreement with regard to the deformation behaviour, but less good agreement with regard to the load capacity. Due to its scope, the use of the described algorithm only seems to make sense for test recalculations or for studies on load-bearing and deformation behavior.

The change in shape of a reinforced concrete component depends on a large number of influencing parameters, which are subject to many coincidences. For practical structural calculations in solid construction, however, it is usually sufficient to know the average curvature behavior of a component in order to be able to calculate the deformation. The stress strain line for concrete shown in Figure 2.4, which includes the usual parabolic rectangle diagram as a special case, can be used as the basis for general calculation methods that can be used to determine shape changes not only in the limit state of the load-bearing capacity, but also under service load and for all intermediate states.

/Figure 2.4 Stress strain relationship/

According to the diagram in Figure 2.4, a course of the □-□ line affine to the pressure range is assumed for the tensile area. The line in the tensile area is determined solely by the parameter ε_{bZ}, so that any calculation value of the concrete flexural tensile strength β_{bZ} can be specified by selecting this elongation value accordingly. A useful value is ε_{bZ} = 0.1 mm/m. Concrete tensile stresses are not taken into account if ε_{bZ} = 0 mm/m is set. With sufficiently large □bZ, unlimited untorn state I can be taken as a basis. If the exponent n = 1 is chosen at the same time, the calculation is carried out as according to the linear theory of elasticity. This is illustrated in Appendix 1, Chapter 1.4.4 in Example 4.

The concrete tensile stresses are not constant in the tensile zone between the cracks, but increase from zero in the crack to a maximum value in the middle between two cracks, which is limited by the flexural tensile strength □bZ, as a result of the bonding effect and the centeriness of the compression zone $_{force}$. Immediately after the crack condition occurs, the mean value of the concrete tensile stresses can be assumed to be equal to half of the flexural tensile strength, whereby the calculated maximum value of the bending tensile force should not be greater than the tensile force absorbed in the crack by the reinforcement.

With increasing elongation in the tensile zone, the bond between reinforcement and concrete is disturbed, so that the mean value of the concrete tensile stresses decreases. This is also the case with increasing tensile zone height, especially if the tensile zone is irregularly enforced with reinforcement. Assuming that the mean value of the tensile stresses falls linearly to zero from the calculated maximum value β_{bz} according to equation (2.2), if the strength in the reinforcement is used in the relevant reinforcement fiber when the yield strength ε_{sS} is reached, the variable mean value of the tensile stresses β_{bZ} is obtained according to equation (2.3). This equation applies to strains of the characteristic fiber (2) in the tensile zone, which are limited by the strain values ε_{bz} and ε_{sS}."

6.8 Limit strains, notes on various standardizations

The limit state of the cross-section is specified by the limit strains:

The following types of stretches are used. The values are only to be understood as an example and can be changed by the user at any time according to the standard used (DIN 1045, EC 2 etc.).

```
Eps.Pressure.B = -3.5000 mm/m (max. compression compression at bending)
Eps.Pressure.centric.B = -2.0000 mm/m (max. compression for centric pressure)
Eps.Tensile strength.B = 100.0000 mm/m (max. tensile strain for concrete)
Eps.Pressure.S = -5.0000 mm/m (max. compression for steel)
Eps.Tensile length S = 25.0000 mm/m (max. tensile elongation for steel)
```

Really important for the limit state are eps.Print.B, eps.Druck.zentr.B and eps.Zug.S. Normally, any limit state of a cross-section can be determined from this without any problems. In exceptional cases, however, the additional strains eps. compression.S and eps. Tension.B. Eps.Compression.S is decisive, for example, if a cross-section is to be calculated only from steel or if the maximum support moment (steel profile lies on the underside) is determined for a composite beam. Eps.Zug.B, on the other hand, is important if there is only one reinforcement point that lies exactly on the polygon edge.

Compressive elongation concrete, tensile elongation reinforcement, each for the design values

DIN 1045 (88) - old edition from 1988

The following values are calculated uniformly for all concretes (B 10 to B 55):

```
Eps.Pressure.B = -3.5 mm/m
Eps.Pressure.center.B = -2.0 mm/m
Eps.S = 5.0 mm/m
```

EC2

The following values are calculated uniformly for concretes C 12/15 to C 50/60:

```
Eps. Pressure.B = -3.5 mm/m
Eps. Pressure.center.B = -2.0 mm/m
Eps. Draw.S = 25.0 mm/m
```

For concretes from C 55/67 onwards, the following limit strains are to be used, each in [mm/m]:

C 55/67	$\varepsilon\upsilon$ = -3.1	$\varepsilon_{centric}$ = -2.2
C 60/75	$\varepsilon\upsilon$ = -2.9	$\varepsilon_{centric}$ = -2.3
C 70/85	$\varepsilon\upsilon$ = -2.7	$\varepsilon_{centric}$ = -2.4
C 80/95	$\varepsilon\upsilon$ = -2.6	$\varepsilon_{centric}$ = -2.5
C 90/105	$\varepsilon\upsilon$ = -2.6	$\varepsilon_{centric}$ = -2.6
C 100/115	$\varepsilon\upsilon$ = -2.6	$\varepsilon_{centric}$ = -2.6

In the case of deformation calculations with the mean values or the characteristic values (in the case of columns) of the material properties, the limit elongation of the respective concretes must be entered correctly. The material characteristics are already included in the material table implemented in INCA2, the corresponding strains can be found in the Excel spreadsheet Baustofftabelle.xls included with the program.

Please also note that the safety coefficient according to the old DIN 1045 standard (1988 edition) depends on the elongation state. Further explanations can be found in the following chapter.

6.9 Reinforced concrete according to DIN 1045, edition 1988 (old standardization)

In order to correspond to the different behaviour of reinforced concrete in the compression fracture of concrete and tensile fracture of reinforcement, a variable safety coefficient was introduced in DIN 1045 (88). If the external stress leads to a brittle failure of the concrete compression zone, a higher level of safety is required than in the case of a ductile failure of the reinforcement (with advance notice).

The safety coefficient thus also takes into account the different dispersions of the parameters of the materials concrete and reinforcement.

6.9.1 Materials according to DIN 1045, edition 1988

Concrete

In the following table, the calculation values for the concrete strength are given for the concrete strength classes. These are to be used with the strains $\varepsilon_{c,centre}$ = - 2.0 mm/m and ε_u = - 3.5 mm/m and an exponent of n = 2. These values are already implemented in the INCA2 predefined material table.

Concrete class	Computational strength βP
B 5	3.5 N/mm²
B 10	7.0 N/mm²
B 15	10.5 N/mm²
B25	17.5 N/mm²
B 35	23.0 N/mm²
B 45	27.0 N/mm²
B 55	30.0 N/mm²

Reinforcing steel BSt 500

The computational strength is f_{yd} = 500 N/mm². The corresponding elongation is ε_y = 2.38095 mm/m, the modulus of elasticity is therefore E = 210,000 N/mm². After yielding, an ideal plastic behaviour is expected, the limit elongation is ε_u = 5.0 mm/m.

6.9.2 Safety concept according to DIN 1045, edition 1988

There is a global, variable safety coefficient. If the stress results in ductile component failure (reinforcement yields), the safety coefficient is γ = 1.75. In the case of a brittle failure (concrete pressure fracture), γ = 2.1 is set. Between these states, linear interpolation takes place in a transition region. This ensures that the greater dispersions of the concrete and the failure mechanism brittle fracture (no prior notice) are offset by a sufficiently high level of safety.

The problem for the calculation in INCA2 is therefore the changing safety coefficient. The consideration would have to be made within the program according to the strain state. However, since INCA2 has not been developed for a specific standard and its specificities, this automatic consideration has been omitted. Rather, it was the author's goal to be able to depict any material behaviour according to any standardization.

6.9.3 Calculation according to DIN 1045 (88)

Method 1 – Safety Case

The cross-section is modelled with the properties of the material shown, and the internal forces are applied as characteristic loads. The normal safety case will be carried out. The resulting safety index γ must be greater than or equal to 1.75 or 2.1, depending on the strain condition. Therefore, in the calculated limit state of the load-bearing capacity, the elongation of the most tensile steel fibre ε_{s2} has to be checked. A distinction is made between the following states:

Elongation $\varepsilon\sigma2$	Required safety factor
ε_{s2} > 3.0 mm/m	γ = 1.75
3.0 mm/m < ε_{s2} < 0.0 mm/m	$\gamma = 1{,}75 + \frac{(3{,}0 - \varepsilon_{s2}) \cdot 0{,}35}{3{,}0}$
ε_{s2} < 0.0 mm/m (completely compressed)	γ = 2.10

If the safety is not sufficient, the cross-section must be improved (reinforcement, dimensions).

Method 2 - Design

In contrast to method 1, the internal forces are calculated with a safety factor of e.g. γ = 1.75. The design is carried out and the elongation ε_{s2} is checked. If a larger safety factor is required, the section properties must be improved accordingly and the design must be repeated.

6.9.4 Comparative calculations with interaction diagrams (e.g. from Schneider construction tables, 12th edition)

The safety factor is already incorporated into these tables and is not considered externally. So you go to the diagrams with the characteristic loads (N_x and My or n and m_y) and you can read out the required reinforcement right away.

Comparative calculation example:

Cross-sectional values:

Circular cross-section ∅ = 1.10 m

d1 = 0.11m d1 / d = 0.10

Ac = $\pi \cdot r^2$ = 0.95 m²

Materials:

Concrete B 25, βP = 17.5 N/mm²

BSt 500

Character. Stress:

M = 1828.75 kNm

N = -2493.75 kN

Related values for the load:

$$m = \frac{M}{A_b \cdot d \cdot \beta_R} = \frac{1{,}82875}{0{,}95 \cdot 1{,}10 \cdot 17{,}5} = 0{,}1$$

$$n = \frac{N}{A_b \cdot \beta_R} = \frac{-2{,}49375}{0{,}95 \cdot 17{,}5} = -0{,}15$$

With these values, the following values can be read from the design diagram for circular cross-sections (Schneider construction tables, page 6.81, issue 12):

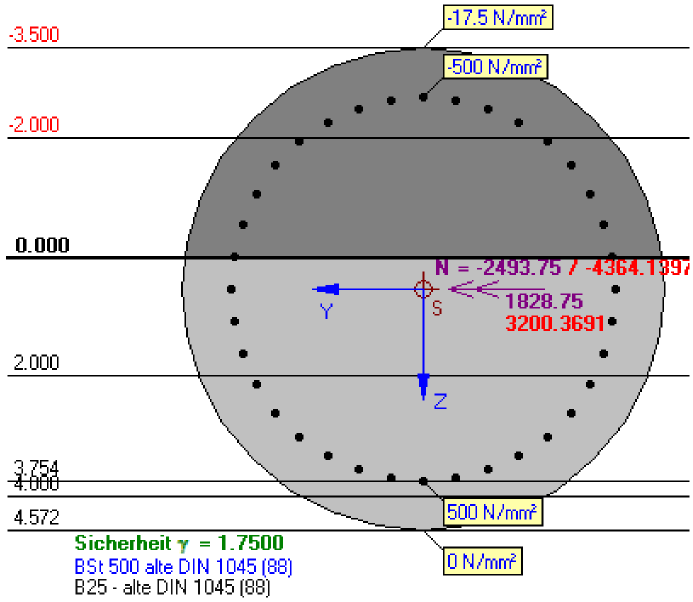
Geometric reinforcement ratio: ω = 0.39

Elongation condition εΧονχρετε = -3.5 mm/m and εΣτεελ = 4.0 mm/m

Reinforcement quantity $A_{s,tot} = \text{tot}\ \omega_0 \cdot \frac{A_c}{\beta_S / \beta_R} = 0{,}39 \cdot \frac{0{,}95}{28{,}6} = 0{,}01296\ \text{m}^2 = 129{,}6\ \text{cm}^2$

It follows from the elongation condition that it is a ductile component failure, as a result of which the safety coefficient γ = 1.75.

A design with the INCA2 program yields a reinforcement quantity of A s.tot = 128.24 cm² with a safety coefficient of ▯ = 1.75. The elongations in the characteristic fibres mentioned above also match very well (εχονχρετε = -3.5 mm/m and εστεελ = 3.822 mm/m). The differences are due to reading inaccuracies.



6.10 Conversion DIN 1045 (Edition 88) and current EC2

Basically, both standards are based on cube and cylinder compressive strengths. However, due to the different storage conditions and test specimen dimensions, the compressive strengths according to the new standard reach lower values than in the old standard. In addition, there are other minor differences in the statistical evaluation of the test series.

In Germany, 150 mm cubes are currently used as test specimens, the measured strength of which corresponds to the second value YY in the designation C XX/YY according to the new standard. Detailed information on this change can be found in the Concrete Calendar, Issue 96, Part 1.

old standardization, using the example of a B 25:

$\beta\Omega N$ = 25 N/mm² Nominal strength (minimum strength after 28 days, 5% fracts)

$\beta\Omega\Sigma$ = 30 N/mm² Series strength (minimum average value, must not be undercut)

The compressive strength test is carried out on the cube with an edge length of 20 cm. Storage takes place for 7 days under water, then in air at 15 - 22°C (see DIN 1045 and 1048). There are conversion factors for different test specimen dimensions (from Betonkalender 1996, p.45):

$\beta\Omega,200 = 0.95 \cdot \beta\Omega,150$

$\beta\Omega,200 = 1.25 \cdot \beta X\psi\lambda,150/300$ for concrete B 5 / 15

$\beta\Omega,200 = 1.18 \cdot \beta X\psi\lambda,150/300$ for concrete B 25 / 35 / 34 / 55

new standardization (EC2), using the example of a C 20/25

fck = 20 N/mm² characteristic strength of the cylinder (Ø 15 cm, h = 30 cm)

fckW = 25 N/mm² Firmness of the cube (15 x 15 cm)

fcm = 28 N/mm² Mean of strength

The compressive strength test is carried out, among other things, on the cube with an edge length of 15 cm. Storage takes place under water for 28 days (see ENV 206 according to ISO 4012-1978). This results in lower compressive strengths than according to the old DIN 1045:

$\beta\chi(I\Sigma O) = 0.92 \cdot \beta\Omega N\ (150\ \mu\mu)$ (only for cubes 150 x 150 !)

Conversion between standards

$$f_{cube,200mm} = \frac{0,92}{0,95} \cdot \beta_{WN,200mm} = 0,97 \cdot \beta_{WN,200mm}$$

Example

fcube = 0.97 · 25 = 24.25 N/mm²

A B 25 according to the old standardization is thus roughly equivalent to a C 20/25 according to the new standardization.

This gives the following table:

Concrete according to old standardization	Conversion $0.97 \cdot \beta\Omega N$	roughly equivalent to concrete according to the new standardization
B 10	9,7	nonexistent
B 15	14,55	C 12/15
B 25	24,25	C 20/25
B 35	33,94	~ C 30/37
B 45	43,65	C 35/45
B 55	53,35	C 45/55

6.11 Materials from DIN 1045 before 1972

When building in existing buildings, it can happen that materials from this period are also used. However, due to the well-known inadequacies of the standard at that time, which eventually led to the improved editions in recent times, it is recommended to adopt equivalent material characteristics according to the new standard. The recalculation of existing structures should therefore be carried out completely in accordance with the new standard.

Material designations according to the pre-1972 standard and their assignment in the new standard

DIN 1045 before 1972		EC 2
Bn 50	fcm = 50 kp/cm²	nonexistent
Bn 80	fcm = 80 kp/cm²	nonexistent
Bn 120	fcm = 120 kp/cm²	nonexistent
Bn 160	fcm = 160 kp/cm²	nonexistent
Bn 225	fcm = 225 kp/cm²	≈ C 12/15
Bn 300	fcm = 300 kp/cm²	≈ C 20/25
Bn 450	fcm = 450 kp/cm²	≈ C 30/37

In the pre-1972 edition of DIN 1045, the concrete grade was characterized by the average 28-day compressive strength of 3 20 cm cubes, which were stored for 7 days moist and 21 days in room air at 15 to 22°C. Individual values were allowed to fall below the nominal strength of the concrete grade by a maximum of 15%. However, due to the more recent findings of the statistics, a reference of the concrete quality to the mean values of the material properties was no longer considered sufficient for building safety. Since any breakage of a component always starts at the weakest point in the area of high stresses, it was required that the design be based on a minimum compressive strength that is reached or exceeded at almost all points of the component.

Thus, a conversion between this old standard and the current standard can be approximated as follows:

> old designation Bn 300 => mean f_{Cm} = 300 kp/cm² = 30 N/mm²
>
> character. Cube compressive strength according to the new standard
>
> fck = f_{Cm} - 8 N/mm² = 30 - 8 = 22 N/mm² (Cube compressive strength)
>
> new concrete about C 20/25 with $f_{ck,cube}$ = 25 N/mm²

However, it should be borne in mind that both post-hardening and concrete damage may have occurred over time in the case of such old structures. The extraction and testing of core boreholes would therefore be desirable in order to achieve an accurate classification according to the new standard. The use of a Schmidt hammer is not recommended, as the concrete surfaces of old buildings often have a much higher strength than the core concrete, which can be decisive for the load-bearing capacity.

It should also be borne in mind that 40 or 50 years ago, concrete could not be produced with the same uniformity as it is today, and that quality monitoring on the construction site was also not as strictly regulated. So, surprises in both positive and negative terms are possible.

6.12 Shrinkage and creep

In INCA2, shrinkage and creep are taken into account at the exact point where these effects take place. Both effects will normally occur at the same time. For a better understanding, shrinkage and creep are discussed separately below.

6.12.1 Shrinkage of concrete

In reality, shrinkage manifests itself in a relative shortening of the concrete in terms of reinforcement. Therefore, it is possible to apply either a tensile pre-expansion (positive) to the concrete or a compression pre-expansion to the steel. In the following example, the concrete shrinks by ε_{sc} = 0.3 mm/m, otherwise there is no further external load, so that the influence of shrinkage can be seen very well.

Cross-sectional parameters:

Concrete C30/37 Average values

BSt 500, averages

w / h / d1 = 40 / 40 / 5 cm

5 Ø 25 mm, $A_{s,dead}$ = 24.54 cm²

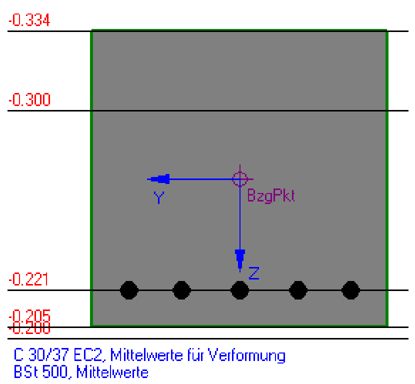
INCA2 files

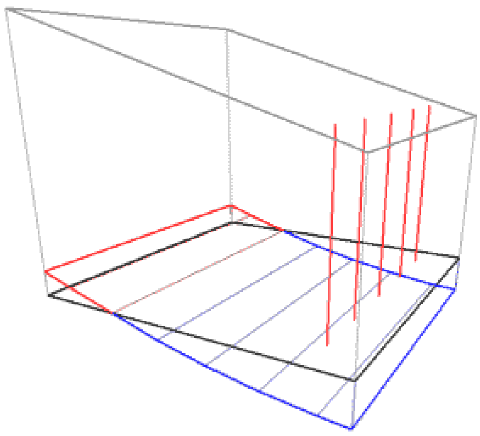
Schwinden_auf_Beton.inc

Schwinden_auf_Stahl.inc

Variant 1 – Shrinkage elongation on the concrete

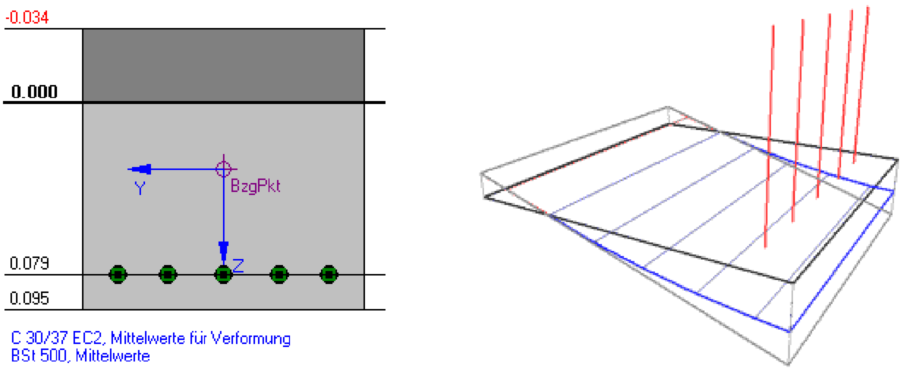
If the concrete undergoes a tensile pre-stretch, the concrete is μενταλλψ πυλλεδ ωιτη εϖ and then firmly connected to the reinforcement. There is now a tensile force in the concrete, so that the concrete would like to deform back, but is prevented from doing so by the reinforcement. As a result, the reinforcement is pressed until a state of equilibrium is established between the concrete tensile force and the steel compressive force.





Variant 2 – Shrinkage elongation on the reinforcement

The second option, to apply a pressure pre-expansion to the steel, works according to the same scheme. The steel is compressed by εϖ and then joined to the concrete. Now there is a compressive force in the steel. The reinforcement would like to deform back, but is prevented from doing so by the concrete. Here, too, a balance of inner forces is restored.



blue = tensile stresses red = compressive stresses

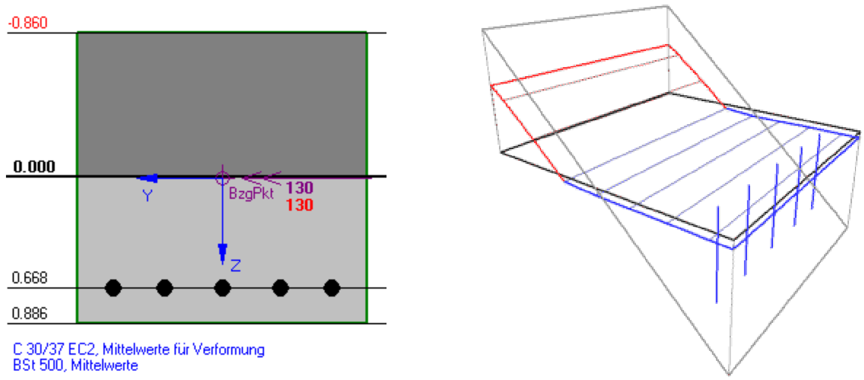
gray = elongation plane black = polygon

The resulting voltage distributions are the same for both approaches. However, the strain plane or the reference point of the strain plane is different because the strain plane is basically related to non-pre-stretched or pre-curved cross-sectional parts. So, in modeling variant 1, this is the steel. This is compressed by the concrete, so that at first glance the strain distribution looks as if the cross-section is completely overprinted. However, it is logical to see that the component's centre of gravity is 0.2710 mm/m shorter. In variant 2, the elongation plane is related to the concrete, so that you can immediately see where compressive and tensile stresses are located in the concrete.

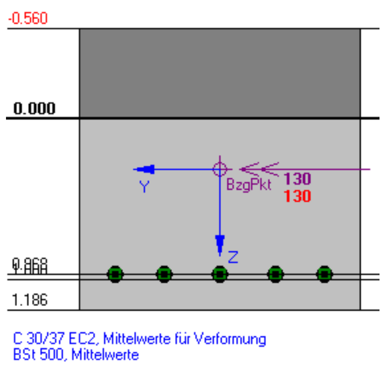
The conversion between the two variants is done simply by adding the shrinkage pre-strain of ε_{cs} = 0.3 mm/m.

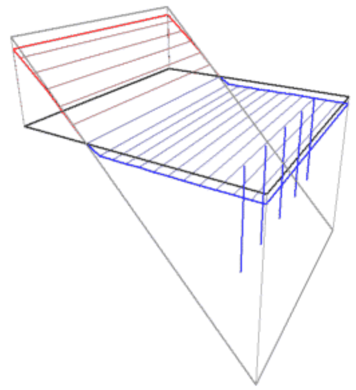
Furthermore, the cross-section is loaded with a bending moment of My = 130 kNm. Again, both variants of modeling are listed.

Shrinkage expansion on the concrete, reference point of the expansion plane is the reinforcement:



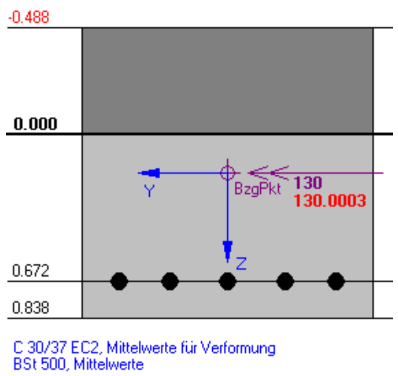
Shrinkage extension applied to the reinforcement, the reference point of the expansion plane is the concrete.

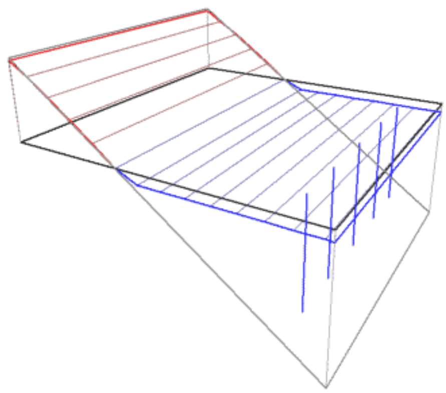




Comparison: without shrinkage

As you can see, the marginal strains and thus the curvatures are smaller than in the cross-sections taking into account shrinkage. Such effects should therefore be taken into account in precise deformation calculations!





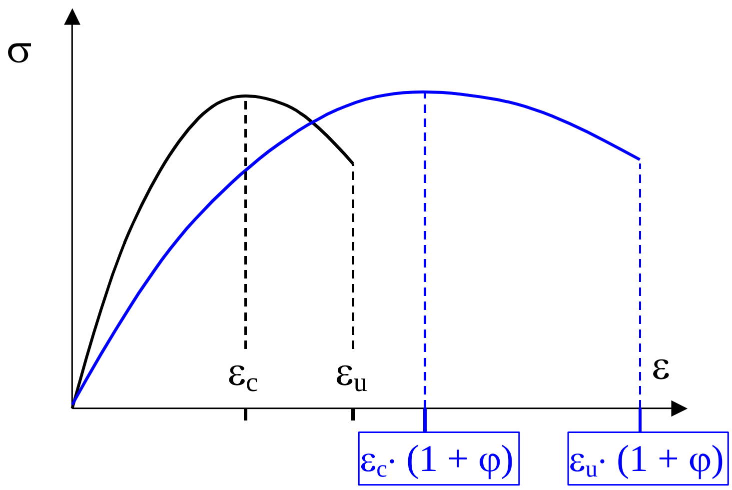
Since in this example the mean values of the material properties were calculated, the stiffening effect of the concrete on tension is taken into account with a fictitious stress expansion line on the tensile side. Depending on the steel elongation, the concrete stress on the tensile side is reduced. Despite the different strain planes, the reduction factor is determined to be the same as 0.6836 in each case.

6.12.2 Creep of concrete

"The creep of the concrete corresponds to an increase in deformations over time, caused by constant stresses at constant temperature and humidity conditions. In the context of practical calculations, creep deformations for stresses under service loads can be measured with sufficient precision by means of a reduced modulus of elasticity. This reduction of the modulus of elasticity is computationally possible with little effort by stretching the stress-strain line with the factor (1 + φ). Corresponding proposals for such a method, which assumes a linear relationship between creep deformations and stresses for the range of low stresses in the working state, can be found, for example, in the CEB/FIB Model Regulation (1987). The assumption of a linear relationship allows creep deformations from different stress components to be superimposed."

[from issue 415 of the DAfStB, Quast, Busjaeger, 1990.]

If the effects of creep on a cross-section are to be investigated, the stress-strain lines for the concrete used must be modified. All strains are to be multiplied by the factor (1 + φ). The limit strains (concrete pressure side) must also be changed accordingly.



The values for the creep coefficient φ can be taken from the standards (e.g. EC2, DIN 1045-1) or calculated more precisely with algorithms printed there.

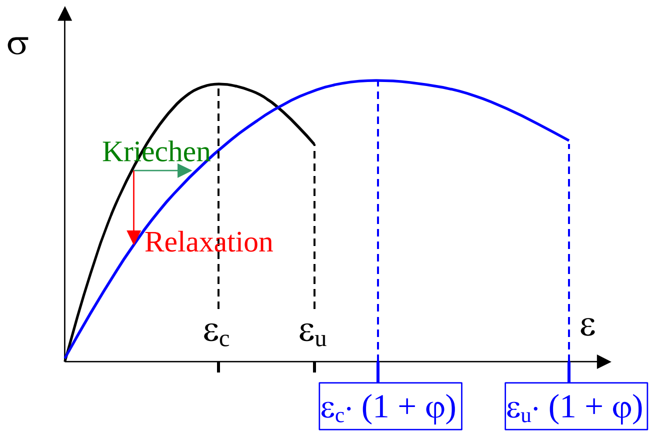
Notes:

The formulation shown assumes that the creep-inducing stress remains constant throughout the period (green arrow in the image below). However, due to the transfer of concrete compressive forces to the compressive reinforcement, the concrete stress will decrease while the bending moment remains the same, so that the relationship shown will slightly overestimate the creep deformations.

More difficult to model is relaxation (red arrow in the following figure), i.e. the reduction of concrete stresses while maintaining constant deformations or strains. This includes, for example, the reduction of coercive forces during the column lowering of a continuous girder. Since the stretching of the stress strain line occurs only in the horizontal direction by a factor (1 + φ), the ratio perpendicular to it is also also (1 + ⍰) only in the case of a linear-elastically defined stress-strain line. If one were to perform a calculation in time steps (corresponding to increments $\Delta\varphi$), the connection for relaxation would be

$$\sigma P\varepsilon\lambda\alpha\xi = \sigma \cdot e^{(-\varphi)}.$$

With the method shown here (stretching with $(1 + \varphi)$), the stresses and thus also the internal forces are overestimated.



However, the greatest uncertainty is generally to determine a correct creep coefficient φ . Since the actual value can easily deviate by 50% or more up or down from the calculated value, a more precise calculation is usually not necessary and makes sense.

Furthermore, it should be noted that the assumed linear creep is only present up to approx. 45% of the concrete strength, at higher stresses the creep influence increases considerably by a non-linear fraction.

7 Computational algorithms

7.1 Voltage integration

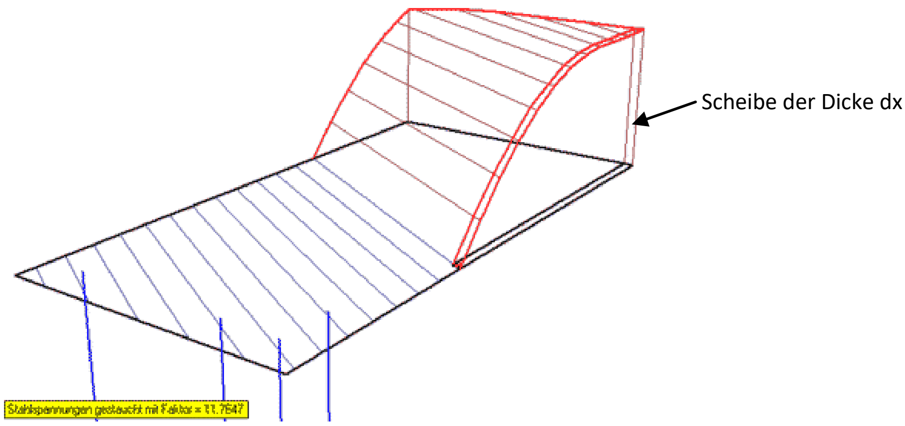
Input values for stress integration are the values for the strain plane $\varepsilon 0$, k_y, and k_z. This means that the current strain and thus the associated stress can be determined at any point in the cross-section during the calculation.

Reinforcement surfaces / individual points:

At the coordinate (y / z) the current elongation (including pre-strain) is determined. This value can be used to calculate the corresponding stress from the material law and then the force.

Polygons

In order to determine the resulting normal force and its point of application, the volume of the stress body must be clearly determined. In the following picture, this is marked in red (concrete pressure area).



As the image shows, the body can be divided into slices of thickness dx. If the course of the tensions as well as the length of this body is known, its volume can be determined:

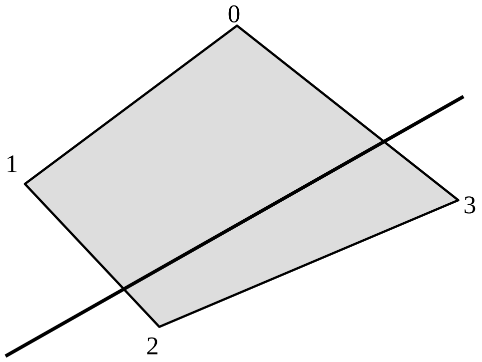
V = Area · dx

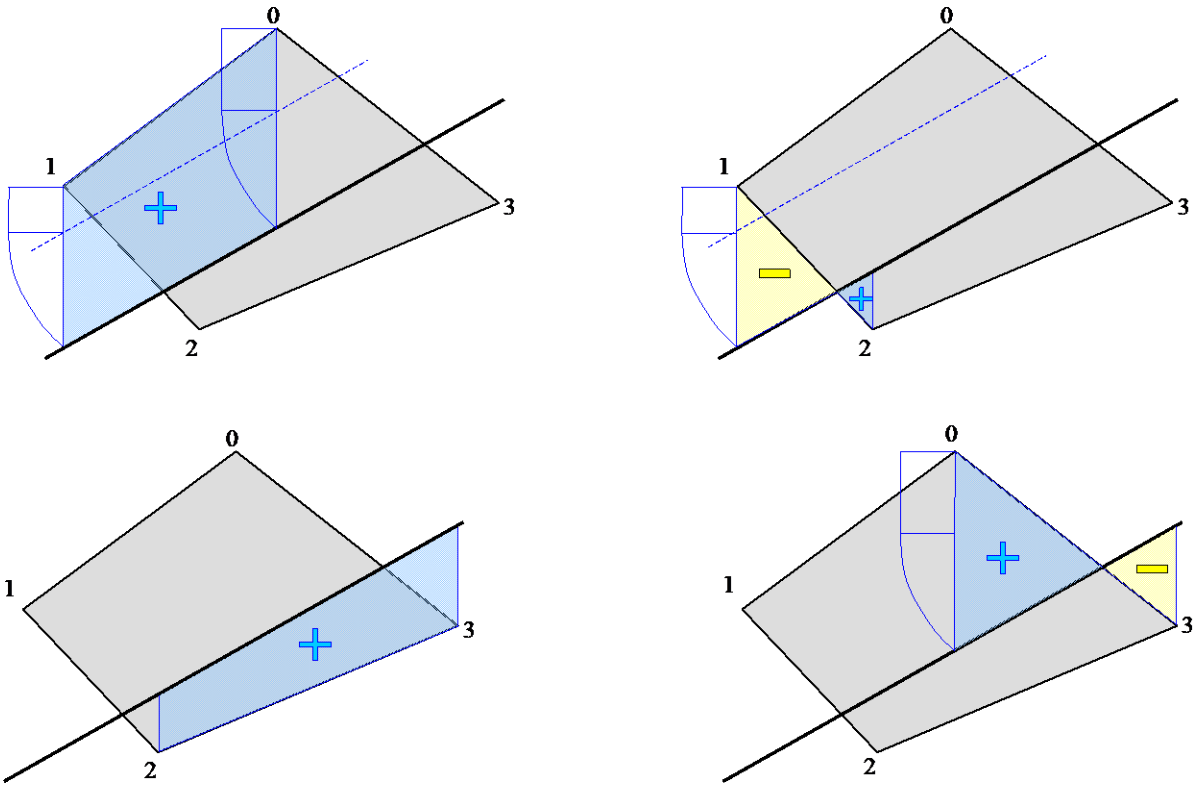
Since the functions that describe the course of the stresses are known and follow simple mathematical functions, the area A can normally be determined analytically (analytic integration).

Subsequently, these volumes of thickness dx all have to be summed up, which is no longer so easy to do analytically. For this reason, numerical integration comes into play here, which is more or less accurate depending on the degree of the basic function.

Basically, the Gauss-Legendre integration is used, which has the advantage of the smaller number of vertices compared to the Newton-Côtes integration. In the linear-elastic material law, only two vertices are required in the transverse direction, in the parabolic rectangular diagram 3 or more (depending on the exponent).

In the case of a polygon, each boundary section is now considered (in the example below 0-1, 1-2, 2-3, 3-0) and the volume is integrated up to a reference line. This reference line can be the y- or z-axis, or even the strain zero line. Rotation in a specific direction (e.g. always orthogonal to DNL) is not required.





If necessary, the edge is divided into several segments. This is particularly useful at points of discontinuity such as the zero line of strain or at points with a change from the parabola to the rectangular part. The sign of the individual faces is mathematically determined "by itself" by the position of the points in relation to the reference line. If the bypass is driven around to the left, there is a positive volume, if the polygon is bypassed to the right, there is a negative volume (e.g. for recesses).

During volume integration, the center of gravity of the individual parts must also be determined. Here, too, the subsequent integration in the transverse direction results in the resulting center of gravity of the stress volume.

7.2 Iteration Procedure

Due to the nonlinear stress strain lines, the strain state for a given combination of internal forces can normally only be determined by iteration. Since voltage integration is very time-consuming, high demands are placed on the iteration method, which should converge quickly and safely even with poor starting values.

A good choice in the case of a constantly differentiable and strictly monotonically increasing function is the Newton method, which is used for safety verification and for determining the strain state.

Procedure elongation condition:

To calculate the strain state, the strain plane must be

$$\varepsilon\,(y,z) = \varepsilon 0 + k_y \cdot y + k_z \cdot z$$

can be varied in such a way that the desired internal forces N x / My / M z are obtained during stress integration over the entire cross-section . This is a three-parameter iteration problem (N_x / My /M_z) = function($\varepsilon 0$, k_y, k_z).

For the calculation of the strain state, the result of a linear calculation is usually used as a starting value. If this iteration does not result in a convergent state, the calculation continues with a damped Newtonian method with the same starting values, in which the step size is initially reduced very strongly, later less strongly. If this procedure does not lead to any results, the safety case is carried out. If a safety of ? > 1.0 is obtained, the result of the safety check is used as the new starting value for the elongation iteration, once again undamped, then damped.

The iteration itself is carried out using the Newtonian method. For this purpose, the derivatives or gradients of the function (N_x / My /M_z) = Fkt($\varepsilon 0$, k_y, k_z) in the individual directions are needed. To do this, the current strain state ($\varepsilon 0$, k_y, k_z) is changed with $\Delta\varepsilon 0$, Δky, Δkz respectively and the resulting values for (N_x / My /M_z) are determined. This can be used to set up the Jakobi matrix and determine the improvement in the iteration value.

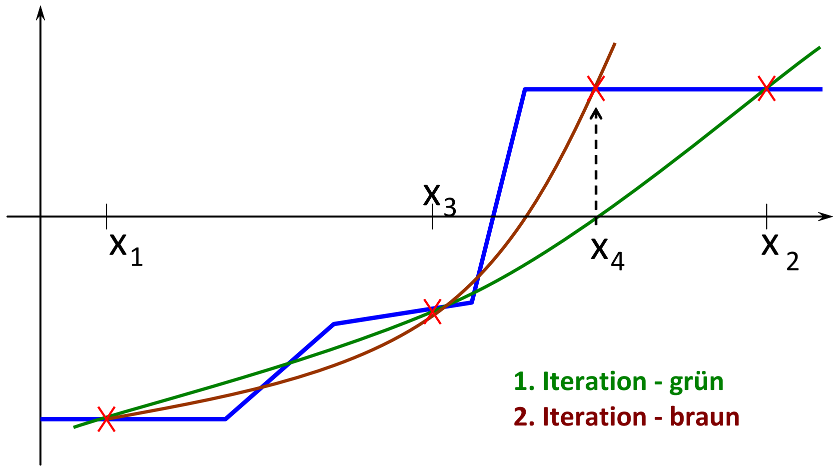
Iteration methods for design, M/k-line and My/M z-line:

In these calculations, only one value needs to be varied to find the zero from the target function. In the case of design, the input value is the factor for the reinforcement, as a result the safety case must provide a safety factor of γ = 1.0. In the case of the M/k line or the My/M_z line, the input value is the strain $\varepsilon 0$ at the coordinate origin, as a result the normal force N_x must correspond to a given value.

In this single-parameter zero search, a method with 3 support points is used, through which a compensatory parabola is laid and the zero point is determined as a new approximation. This method has the advantage of very fast and, above all, safe convergence, even with extremely poor starting values. Horizontal curves in the functional curve of the parameter $\varepsilon 0$, as they can occur when calculating the M/k line or My / M_z line, are not problematic. The Newtonian method would fail in such a case. The disadvantage, however, is that a starting value on the positive side and a starting value on the negative side must be known.

Scheme of iteration:

1. Select seed value x1 and x2 and calculate function values, condition: $fkt(x_1) < 0$
   $fkt(x_2) > 0$
2. for x3 = 1/2 · (x1 + x2) choose the next value and calculate $fkt(x_3)$
3. Place the parabola through these three points and calculate zero => x4
4. $fkt(x_4)$ gives a new point for the parabola, continue at point 2



The curve shown here is an extreme example, but it shows the stability and speed of the algorithm very clearly. In particular, the Newton method would fail here due to the horizontal sections. These horizontal areas result, for example, from the elastic-plastic definition of the material laws. If the reinforcement is stretched even further in the plastic state, there are no differences in stress, so that the normal force remains constant.

7.3 Limit Strains / Strain Indicators (DKZ)

7.3.1 Limit strains

The limit state of the cross-section is specified by the limit strains. The following types of stretches are used. The values are only to be understood as an example and can be changed by the user at any time according to the standard used (DIN 1045, EC 2 etc.).

```
Eps. Pressure.B = -3.5000 mm/m (max. compression compression at bending)
Eps. Pressure.centric B = -2.0000 mm/m (max. compression compression at centric pressure)
Eps. Tensile length B = 100.0000 mm/m (max. tensile elongation for concrete)
Eps. Pressure.S = -5.0000 mm/m (max. compression compression for steel)
Eps. Tensile length S = 25.0000 mm/m (max. tensile elongation for steel)
```

Essential for the limit state are actually only eps. Print.B, eps. Druck.zentr.B and eps. Zug.S. Normally, any limit state of a cross-section can be determined from this without any problems. In exceptional cases, however, the additional strains eps. Print.S and eps. Zug.B. Eps.Druck.S is decisive, for example, if a cross-section is to be calculated only from steel or if the maximum column torque (steel profile lies on the underside) is determined for a composite girder. Eps.Zug.B, on the other hand, is important if there is only one reinforcement point that is exactly on the polygon edge (e.g. glued reinforcement).

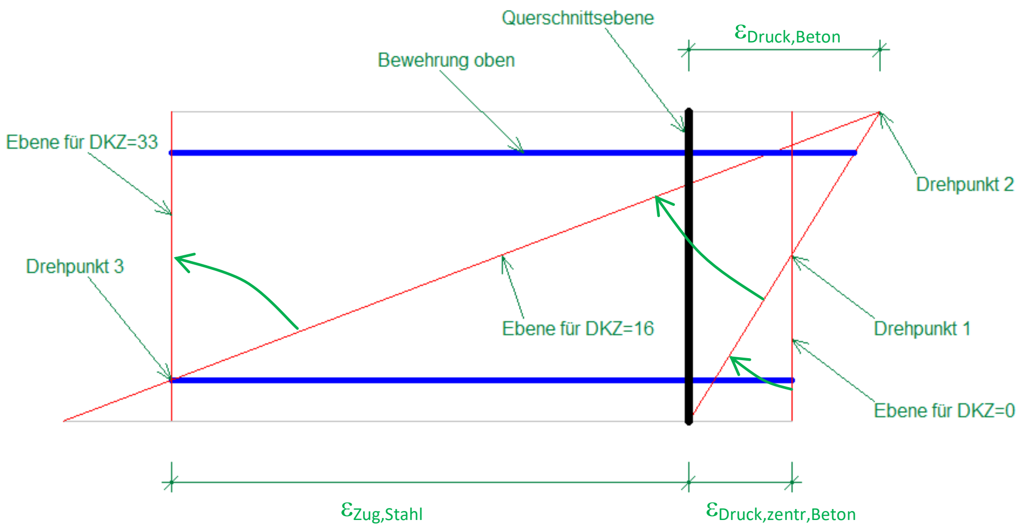
However, by selecting a specific limit strain, tests can also be carried out in the state of use. For example, it is interesting to know for which bending moment the stress in the steel has a certain value, e.g. in order to perform a simplified crack width verification.

For the design of a reinforced concrete cross-section, the design values of the material properties are used. According to the current standardization (EC2), the limit strain for concretes C12/15 to C 50/60 is eps. Pressure.B = -3.5 mm/m. The high-strength concretes C55/67 to C 100/115 have a smaller limit elongation for the design, which must be entered in the corresponding field when using such concrete! Even in deformation calculations with the mean values or the characteristic values (in the case of columns) of the material properties, the limit elongation of the respective concretes must be entered correctly. The material characteristics with the corresponding strains can be found in the Excel spreadsheet Baustofftabelle.xls included with the program.

7.3.2 DKZ - Strain Indicator

The strain index is an artificially introduced value to make some calculations (e.g. safety proof) a little easier. The range of DKZ runs from 0 to 33, where DKZ = 0 represents centric pressure and DKZ = 33 centric tension with the maximum allowable strain. Each DKZ is assigned exactly to a limit strain state, whereby all intermediate values such as DKZ = 2.7854 are also possible. The following elongation planes are passed:

1. Centrically pressed, DKZ = 0 (centrically pressed concrete cross-section)
2. Rotation of the strain plane around pivot point 1 until the limit strain $\varepsilon_{Pressure,concrete}$ is reached on the upper side and straight $\varepsilon = 0$ on the lower side
3. Rotation of the elongation plane about pivot point 2 until the elongation Tensile,steel is reached on the underside in the position of the lower reinforcement , corresponds to DKZ = 16
4. Rotation of the elongation plane by pivot point 3 until the elongation $\varepsilon_{Tensile,steel}$ is also reached on the upper side , corresponds to DKZ = 33



Application Safety Proof:

In the safety case for the internal forces combination N_x / My / M_z , a factor is sought by which all three values are to be multiplied in order to reach the limit state. The angles between My and M_z as well as N with the plane My / M_z remain the same and are therefore the target values to be determined during iteration. Input values (or variable quantities) are the main direction of curvature and the DKZ, which always gives a limit strain state.

All in all, this value is rather uninteresting for the average user. For programming or to get a deeper insight into the calculation or possible convergence difficulties, it is an important program item in INCA2.

Further information can also be found in the booklet by Busjaeger and Quast (1990), which describe in detail the computational routines for the program MasQue (a precursor of INCA2).

8 Examples

In the following examples, the operation of the INCA2 program is explained as well as methods and possibilities for modeling INCA2 cross-sections are presented.

The option of recesses for the reinforcing bars was added to the INCA2 program a little later. The examples shown below have been calculated without this option, so there will be slight differences in the results. If necessary, deselect this item in the menu Tools => Settings => Results, so that the calculated results match the examples presented here.

Calculations with the programs INCA2 and ABaS

Other possible applications

8.1 Example 1 – Simple rectangular cross-section

Start INCA2 or select New from the File menu to get an empty record. As standard, the material characteristics for C20/25, C30/37, C40/50 and reinforcing steel B 500 are already available as design values.

To create the rectangular concrete cross-section, select ***the Rectangle sub-item from the Input*** menu. In the window that appears, please enter the corner coordinates of the rectangle (diagonally opposite). The information is given in meters, and both a period and a comma are accepted as separators for the decimal places.

```
Corner 1:     y = -0.20     z = -0.30
Corner 2:     y = +0.20     z = +0.30
```

Continue to select ***C30/37 PR as the material, design values***. If the ***Cross-section*** option is clicked, you can confirm with OK.

To create the reinforcing bars, choose ***the Point sub-item from the Input*** menu. In the window that appears, select the ***Reinforcement Point option*** and in the selected box for Material, select the B 500 reinforcing steel. For the time being, the diameter is set at 10 mm.

Then create four reinforcement points by entering the y/z coordinates and confirming them with the ***Apply*** button . Please make sure to press the ***Apply*** button only once at a time, otherwise two reinforcing bars can be created on top of each other.

```
Point 1       y = +0.15     z = +0.25
Point 2       y = +0.05     z = +0.25
Point 3       y = - 0.05    z = +0.25
Point 4       y = - 0.15    z = +0.25
```

Then select the Cancel button to close the window.

In addition, a load is applied to the cross-section. To do this, select ***the sub-item*** Effects in the ***Input menu*** and enter the value for the bending moment My:

```
My = 450 kNm
```

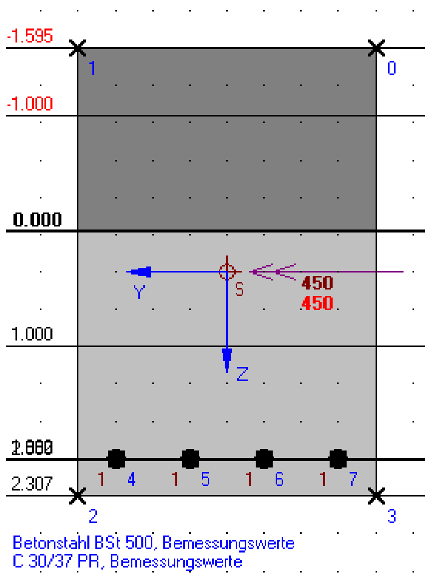
Now please check whether the limit strains are set correctly. To do this, select ***the sub-item*** Limit strains in the ***Input*** menu and click on the ***EC2*** button, then on ***OK.*** It is important that the maximum compressive elongation for the concrete is -3.5 mm/m and the maximum tensile elongation of the reinforcement (steel) is 25.0 mm/m.

By selecting the menu item Results, sub-item ***Strain state***, you can perform the strain calculation for this cross-section. Due to the very small reinforcement chosen, the bending moment of My = 450 kNm cannot be absorbed. The bending moment that can currently be absorbed is M_y = 73.44 kNm.

In order to give the cross-section a sufficient load-bearing capacity, the reinforcement is increased. To do this, select ***the Design sub-item in the Results menu*** and then the first option All Reinforcement Groups in the window that opens. After clicking on the ***OK*** button, a message box appears stating that the reinforcement must be increased by a factor of 6.8778 in order to be able to absorb the load.

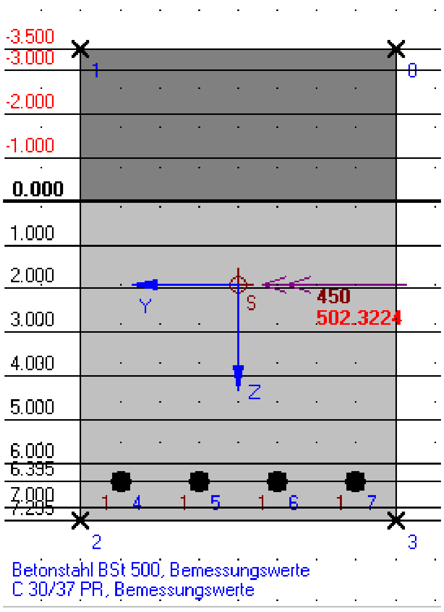
The reinforcement is automatically adjusted and the strain calculation is performed. The upper edge of the cross-section has an elongation of -3.5 mm/m and is therefore maximally compressed.

Next, mark all four reinforcing bars on the underside of the cross-section. To do this, click on them one after the other with the mouse and hold down the ***Ctrl*** key. Then select ***Properties*** from the ***Edit*** menu. The reinforcing bars currently have a diameter of 26.2 mm. Increase this to 28 mm and confirm with ***OK***. When calculating the strain again, it can be seen that the cross-section is no longer subjected to so much stress, the maximum compressive elongation is - 1.595 mm/m. The following image shows this elongation state. The upper, dark hatched area represents the pressure zone, the red numerical values on the left edge are the corresponding pressure strains. The blue numbers at the bottom right of the points are the node numbers, the numbers shown in brown at the bottom left of the reinforcing bars indicate the reinforcement group. Furthermore, in the lower left corner, the used materials are displayed.



Now check how much the stress can be increased by selecting ***Safety*** Verification ***from the Results*** menu . In the following message box, the message appears that the load (in this case My) can be increased by a factor of 1.1163 until the load capacity is reached. This corresponds to a bending moment of My = 502.3 kNm.

The bending moment vector is labeled with two numerical values. The upper value (shown in brown) is the load entered by the user. The lower value (red) is the result of the current calculation, i.e. the maximum load in this case.

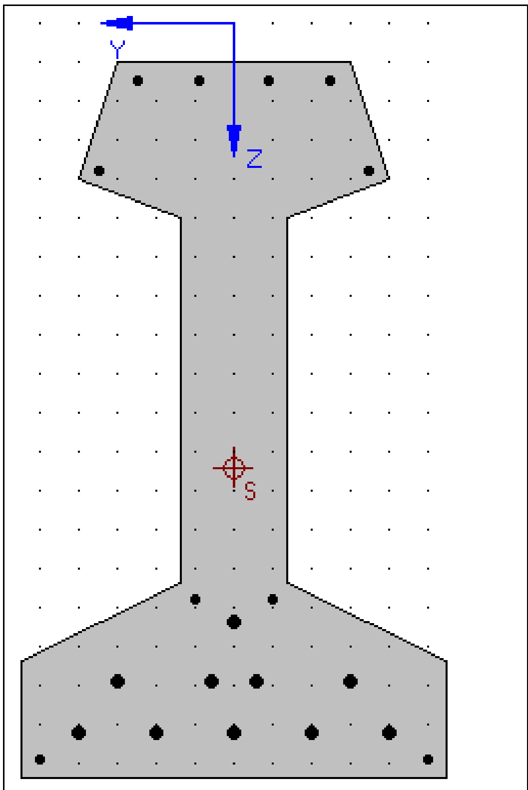


8.2 Example 2 – Prestressed concrete beam

Start INCA2 or select New from the File menu to get an empty record. As standard, the material characteristics for C20/25, C30/37, C40/50 and reinforcing steel B 500 are already available as design values. The prestressing steel required in this example can be imported from a predefined material table. To do this, select ***the sub-item*** Materials ***in the Input menu and the*** Materials Table ***button in the window*** that opens. In the first part of the following table, the materials for tensile reinforcement are summarized. Mark the ***prestressing steel 1550/1770, design values*** and apply it. The list of materials can then be closed.

The cross-section used in this example comes from a bridge north of Hamburg and was already made in the 1960s. In 2002, due to uncertainties in the tendons, the cross-section had to be examined more closely. However, the material properties (as well as some dimensions) used in the following example have been simplified.

In the Input menu, select the sub-item Points as a list to create the concrete cross-section. Use the tab tab Free Table and copy the following pairs of values (y / z) into the box:



Coordinates

y	z
0.1500	0.0500
0.2000	0.2000
0.0700	0.2500
0.0700	0.7200
0.2750	0.8200
0.2750	0.9700
-0.2750	0.9700
-0.2750	0.8200
-0.0700	0.7200
-0.0700	0.2500
-0.2000	0.2000
-0.1500	0.0500

Then select New Polygon ***as the type of points***, C40/50 PR ***as the material, design values*** and confirm with ***OK***.

Second, the slack reinforcement is defined. To do this, select ***the sub-item*** Points as a list ***in the Input menu***, as before. Now, however, select the ***Reinforcement Point option*** and the material is the ***reinforcing steel B 500, Design Values***. The reinforcement group is 1, set the diameter to 10 mm. The coordinates (y / z) can be found in the following list:

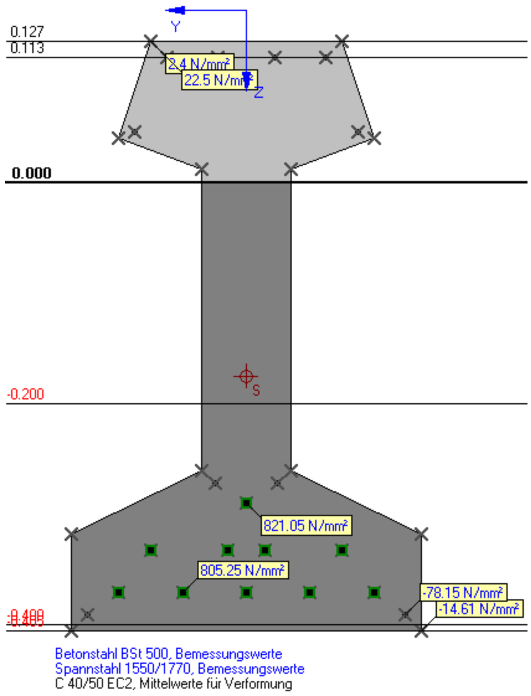
```
 0.0500      0.7400
-0.0500      0.7400
 0.2500      0.9450
-0.2500      0.9450
 0.1250      0.0750
-0.1250      0.0750
 0.0450      0.0750
-0.0450      0.0750
 0.1750      0.1900
-0.1750      0.1900
```

Thirdly, the prestressed reinforcement that was installed in the precast plant in the precast bed is defined. Please make sure that you choose a ***new reinforcement group 2 and*** prestressing steel 1550/1770 ***as the material*** . The diameter is now 16 mm. Furthermore, a ***pre-elongation*** must be defined by clicking on the ***Pre-elongation/Pre-curvature*** button. A new window will open in which you enter the pre-elongation ***eps.0 = 4.5 mm/m*** and confirm with ***OK***. The coordinates (y / z) can be found in the following list:

```
 0.2000      0.9100
 0.1000      0.9100
 0.0000      0.9100
-0.1000      0.9100
-0.2000      0.9100
 0.1500      0.8450
 0.0300      0.8450
-0.0300      0.8450
-0.1500      0.8450
 0.0000      0.7700
```

Now select ***the sub-item*** Strain condition in the ***Results*** menu to see the stresses in the precast plant after the tendon anchorage has been cut. The pressure zone is located at the bottom, on the upper side the cross-section is wide open. It should be noted, however, that the design values were currently calculated!

If the cross-section is calculated with the mean values of the material properties (i.e. the existing condition in the precast plant), then the cross-section is subjected to tension on the upper side, but could still remain in the uncracked state. To do this, import ***the concrete*** C40/50 EC2, average values for deformation***, in the menu item Input***, sub-item ***Materials***, button Material ***Table***. Then change the material of the cross-section (polygon) by selecting it (at the edge) and ***setting the newly imported C40/50 average values as the current material in the Edit*** menu, sub-item ***Properties***. A new calculation of the elongation condition results in the following picture:



To continue with example 3 (subsequently added cross-section), reset the material property of the polygon back to the concrete C40/50 PR, design values.

Notes on preload in the fitted bed:

The prestressed concrete girder modelled here was prestressed in the prestressed bed (in the pre-cast plant). In a first step, the prestressing steel is installed in the tension bed and prestressed, then the concrete is laid stress-free. After the concrete has hardened, the prestressing steel is separated from the tension bed and the concrete is compressed. In this process, the original elongation of the prestressing steel is reduced by the extent of the compressive elongation of the concrete.

In the definition of prestressing steel reinforcement, a pre-elongation of $\varepsilon\pi$ = 4.5 mm/m was entered. After calculating the cross-section without external loading, the elongation is reduced by the amount of concrete compression. Depending on the position in the cross-section, this results in residual prestressing steel strains of 4.13 mm/m to 4.21 mm/m.

```
Reinforcement group No. 2
--------------------------------------------------------------------------------------------------
   Material: Prestressing steel 1550/1770, design values (steel)
   Pre-elongation: eps = 4.5000 + y * 0.0000 + z * 0.0000
   Points (10 pcs.)   Coordinates (Y / Z) Area [cm²] eps [mm/m] Sigma[N/mm²] Force [kN]
       1  ( 22)       0.2000 /     0.9100        2.0106        4.1295      805.2532      161.9057 *
       2  ( 23)       0.1000 /     0.9100        2.0106        4.1295      805.2532      161.9057 *
       3  ( 24)       0.0000 /     0.9100        2.0106        4.1295      805.2532      161.9057 *
       4  ( 25)      -0.1000 /     0.9100        2.0106        4.1295      805.2532      161.9057 *
       5  ( 26)      -0.2000 /     0.9100        2.0106        4.1295      805.2532      161.9057 *
       6  ( 27)       0.1500 /     0.8450        2.0106        4.1671      812.5851      163.3799 *
       7  ( 28)       0.0300 /     0.8450        2.0106        4.1671      812.5851      163.3799 *
       8  ( 29)      -0.0300 /     0.8450        2.0106        4.1671      812.5851      163.3799 *
       9  ( 30)      -0.1500 /     0.8450        2.0106        4.1671      812.5851      163.3799 *
      10  ( 31)       0.0000 /     0.7700        2.0106        4.2105      821.0451      165.0809 *
--------------------------------------------------------------------------------------------------
                                      A.s,ges = 20.1062 cm² F.s,ges = 1628.1292
      F.Train = 1628.129 kN / s.y = 0 m / s.z = 0.87 m
```

The calculation method in INCA2 thus corresponds to the prestressing in the prestressing bed, in which the prestressing steel is connected to the stress-free concrete after prestressing and an equilibrium is achieved between the concrete and the prestressing steel by compressing the concrete.

Cross-sections with subsequent bonding must therefore be modelled with adapted prestress.

Note for prestressing with subsequent bonding :

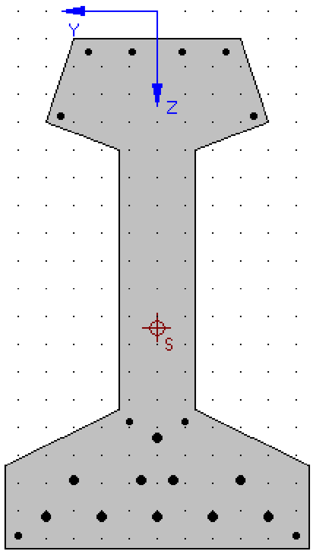
In the case of post-tensioning with subsequent bonding, the structural engineer defines a prestressing to be applied. This prestressing is applied by a tensioning press on the construction site and the concrete is pressed at the same time as it is tensioned. This means that the prestressing applied on the construction site already corresponds to the value resulting from the cross-section in equilibrium.

In INCA2, therefore, a prestressing must be defined that corresponds to the desired prestressing plus the concrete compression compression. In the case of simple cross-sections, this can basically be calculated manually beforehand. However, it is also possible to adjust the preload iteratively, so that after calculating the elongation state, the desired value of the preload is obtained.

8.3 Example 3 – Subsequently added bridge cross-section

This example builds on the cross-section generated in Example 2. Create the prestressed precast beam explained there and then proceed from this point.

Cross-section from example 2:



The girder is transported to the construction site and installed as a single-span girder with a span of 16 m. The load due to 1.0 times its own weight is

g = 25 kN/m³ · 0.2488 m² = 6.22 kN/m

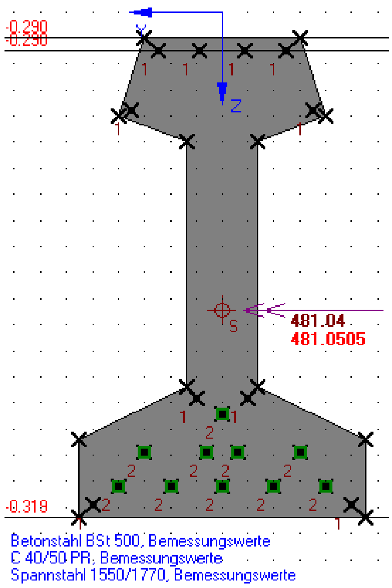
The additional concreting load is (area of the supplemented cross-section 0.3525 m²)

g = 25 kN/m³ · 0.3525 m² = 8.8125 kN/m

The resulting bending moment is thus

$$M = \frac{(g_1 + g_2) \cdot \lambda^2}{8} = 481{,}04 \text{ kNm}$$

With this bending moment, the precast beam is loaded during concreting without additional support. The fresh concrete does not yet absorb stresses. For the precast beam, the following completely overpressed condition results in the middle of the field:



To do this, take a look at the ***3D stress state*** in the ***Results menu, and display*** the ***numerical results*** in the Results menu item . The calculation resulted in an elongation state with the following elongation plane:

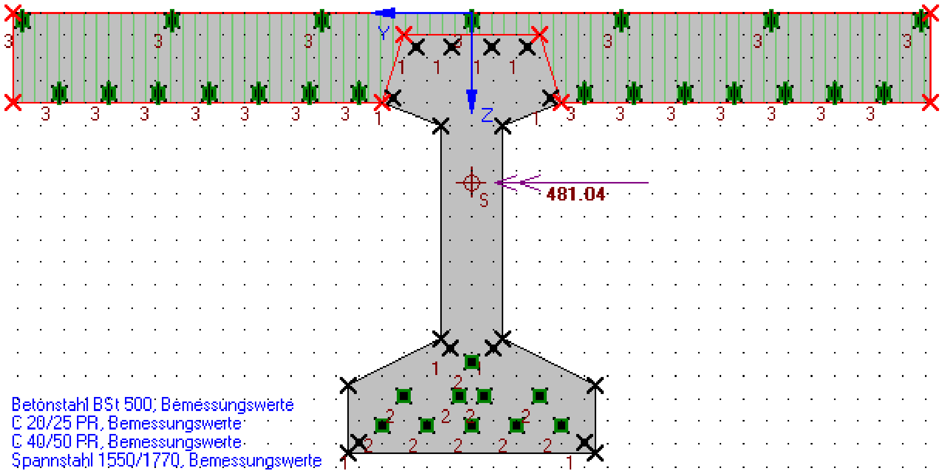
```
eps.0 = -0.2879 mm/m      deps/dy = 0        deps/dz = -0.0315 mm/m
```

Now the subsequently added plate is modeled. A C20/25 was used as the material on the construction site. To create the plate, the first step is to create the four additional vertices, but without creating the polygon at the same time. To do this, select ***Points as a list*** and select ***Polygon Points***.

```
 1.0125    0.0000
 1.0125    0.2000
-1.0125    0.2000
-1.0125    0.0000
```

Then the points must be connected to form a polygon. To do this, select ***Polygon*** from the ***Input menu***. After selecting the material ***C20/25 PR, design values*** and confirming the ***Create Polygon button with the mouse***, click on the vertices of the area to be added later. When the polygon is closed, the cross-section looks like this (points and polygon marked here in red).

Supplemented cross-section, here already shown with reinforcement and pre-extension:



However, the current modelling did not take into account the fact that the slab was concreted later, namely after the precast beam had already been stressed and had undergone compression compression on the top. In order to take this effect into account, a ***pre-expansion / pre-curvature*** must be applied to the supplemented cross-sectional part . To do this, select the polygon at the edge and display the ***properties***. Select the ***Pre-Expansion / Pre-Curvature button*** and enter the elongation plane previously calculated for the concreting load with a different sign in each case:

```
eps.0 =  +0.2879 mm/m      deps/dy = 0          deps/dz = +0.0315 mm/m
```

If you now perform the calculation of the supplemented cross-section again with My = 481.04 kNm, the result is the same as in the previous calculation of the finished beam (small deviations in the 4th decimal place are due to the input with only 4 significant digits). Especially in the 3D voltage state, you can see that the subsequently added plate does not yet have any voltages.

In addition, the reinforcement of the slab must now be modelled. To do this, select the menu item ***Points as a list***, the option ***Reinforcement points***, reinforcement group 3, diameter 14 mm and the reinforcing steel BSt 500, design values as the material. For pre-elongation / pre-curvature, enter the previously mentioned strain plane here as well.

Points for reinforcement:

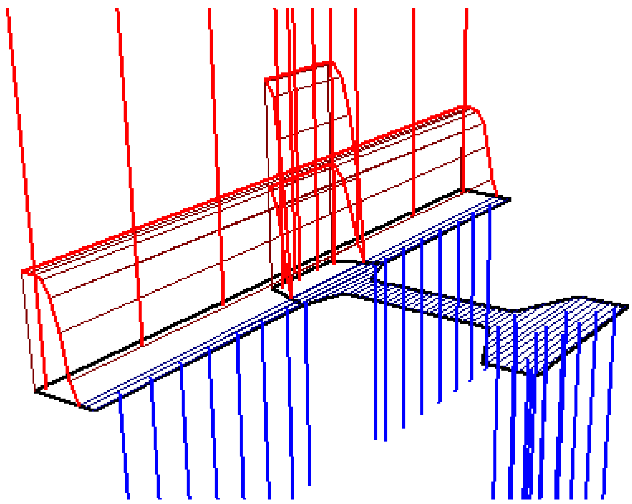
```
 0.0000    0.0200
 0.3300    0.0200
-0.3300    0.0200
 0.6600    0.0200
-0.6600    0.0200
 0.9900    0.0200
-0.9900    0.0200
 0.2500    0.1800
-0.2500    0.1800
 0.3600    0.1800
-0.3600    0.1800
 0.4700    0.1800
-0.4700    0.1800
 0.5800    0.1800
-0.5800    0.1800
 0.6900    0.1800
-0.6900    0.1800
 0.8000    0.1800
-0.8000    0.1800
 0.9100    0.1800
-0.9100    0.1800
```

After complete modelling of the subsequently added cross-section, check its correctness by calculating the strain condition for My = 481.04 kNm. There must still be no differences to the previous invoices.

To determine the maximum bending moment that can be absorbed by the cross-section, select ***the safety check sub-item in the Results menu***.

max M_y = 2434.7 kNm

By looking at the 3D stress state, the load-bearing behavior and the different stresses in the cross-section can be easily recorded.



Note that despite the max. compression compression of -3.5 mm/m shown in the cross-sectional image, the concrete edge is only compressed at -3.212 mm/m. Have a look at the detailed numerical results, which you can also find in the Results menu . An excerpt from this is shown below.

```
Polygon No. 2
--------------------------------------------------------------------
   Material : C 20/25 PR, Design values (concrete)
   Pre-elongation: eps = 0.2879 + y * 0.0000 + z * 0.0315
   Reduction factor for concrete tensile zone = 0.0000
   Points (8 pcs.) - Coordinates (Y / Z) eps [mm/m] Sigma[N/mm²]
       1  (32)       1.0125 /     0.0000       -3.2121      -11.3333
       2  (33)       1.0125 /     0.2000        1.4716        0.0000
       3  ( 1)       0.2000 /     0.2000        1.4716        0.0000
       4  ( 0)       0.1500 /     0.0500       -2.0412      -11.3333
       5  (11)      -0.1500 /     0.0500       -2.0412      -11.3333
       6  (10)      -0.2000 /     0.2000        1.4716        0.0000
       7  (34)      -1.0125 /     0.2000        1.4716        0.0000
       8  (35)      -1.0125 /     0.0000       -3.2121      -11.3333
```

As a result, the concrete could be pressed slightly harder, but this will not lead to a noticeable increase in load-bearing capacity.

The calculations shown here were carried out exclusively with the design values of the material properties. This does not determine the actual state, but a limit state that normally never occurs.

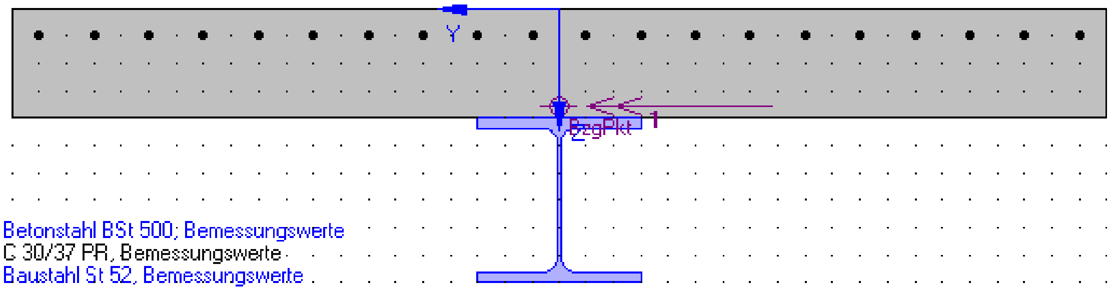
In order to get an insight into how great the real, existing stresses are, the calculation must be carried out with the average values of the material properties (especially for the concrete). This makes it possible to determine how high the stress on the pressed tensile zone is, whether and how far the cross-section tears open and, of course, how great the subsequent compressive stresses are in the panel and finished part. Furthermore, it is possible to estimate the magnitude of the cracks that occur.

Due to creep and different shrinkage behavior of the cross-sectional parts, stresses will be redistributed over time. All in all, this will reduce the preload. However, how the pressure zone of the finished part or the plate behaves in relation to the distribution and magnitude of the compressive stresses can only be shown by a detailed calculation with shrinkage and creep.

8.4 Example 4 – Composite beams

Start INCA2 or select ***New*** from the ***File menu*** to get an empty record. As standard, the material characteristics for C20/25, C30/37, C40/50 and reinforcing steel B 500 are already available as design values. The structural steel S355 required in this example can be imported from a predefined material table. To do this, select the sub-item Materials in the Input menu ***and the*** Materials Table ***button in the window*** that opens. The first part of the following table summarises the materials for steel. Mark the ***S355 structural steel, design values*** and apply it. The list of materials can then be closed.

The following composite cross-section is to be generated:



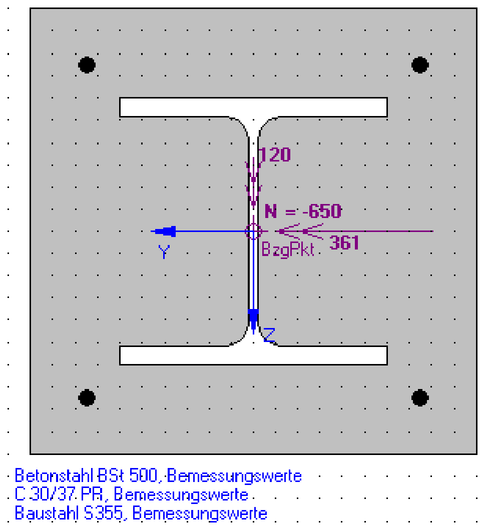
To do this, first create the concrete slab (w / h = 2.0 / 0.2 [m]) with a C30/37 design values. Make sure that the top edge of the plate is at z = 0 and the vertices are at +/1.0 m.

Then create exactly one reinforcement point (B 500) at y = 0.95 m / z = 0.05 m, diameter 16 mm. Then select it and choose ***Slide*** from the ***Edit*** menu. For relative displacement, εντερ Δψ = –0.10 μ / Δζ = 0.00 μ, σελεχτ Χοπψ, and then select ***19 copies***.

Thirdly, a HEB 300 is created at the bottom of the plate. To do this, in the ***Input menu, select*** the sub-item ***Complete-Cross-Sections***, tab I-Profiles. In the window that opens, the profile type and dimensions (HEB 300) must be defined. In addition, select S ***355 structural steel as the material***, design values and y = 0.0 m / z = 0.35 m as the position of the center of gravity.

After entering the load My = 1.0 kNm, the maximum load capacity can be determined with the safety check to M_y = 1446 kNm.

As a second example, the following composite column will be created. The outer dimensions are w / h = 40 / 40 cm, inside a HEB 240 was concreted in. As before, the materials are C 30/37, B 500 and S355.



In a first step, create the polygon (40 x 40 [cm]) for the concrete cross-section with the center of gravity at the coordinate origin. Then create the 4 reinforcing bars in the corners, each with 5 cm edge spacing and a diameter of 16 mm.

Next, create the HEB 240 profile (structural steel S355). This time, the center of gravity of the profile should be positioned at y = 0 / z = 0. This means that the profile comes to rest exactly in the middle of the support.

However, the problem with the previous modelling and subsequent calculation is that both concrete and steel are now assumed by the programme at the point of the profile. The area of the concrete polygon is therefore assumed to be too large when calculating the stresses.

For this reason, it is necessary to define a recess in the concrete of the same size at the location of the HEB 240 profile. To do this, open the window to create an I profile a second time-. Choose the same size (HEB 240), but this time the material is the concrete of the surrounding polygon (C 30/37) and also the recess option.

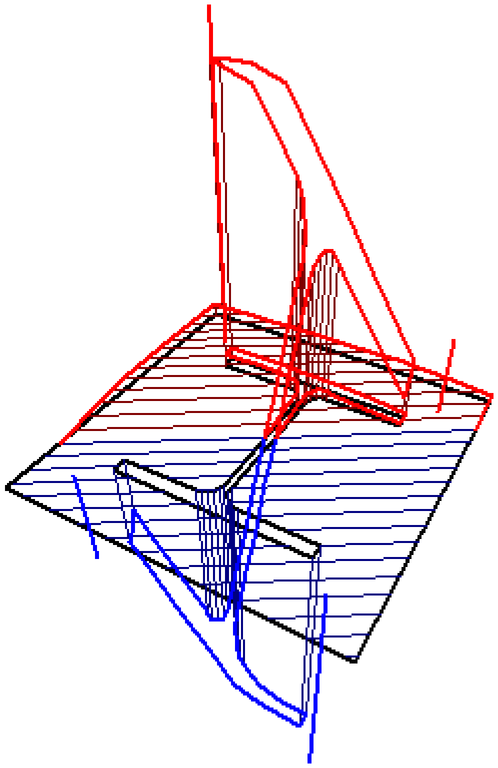
This ensures that the stresses of the concrete in this area are subtracted again later during the calculation and thus "too much" area is not taken into account.

Then enter the following charge

Nx = -650 kN My = 361 kNm Mz = 120 kNm

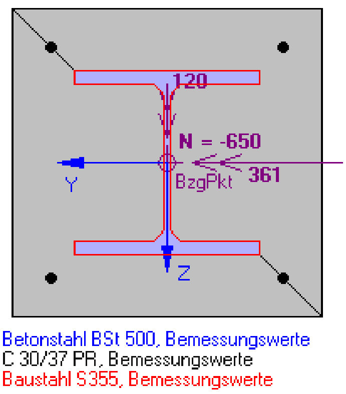
and the elongation condition can be calculated.

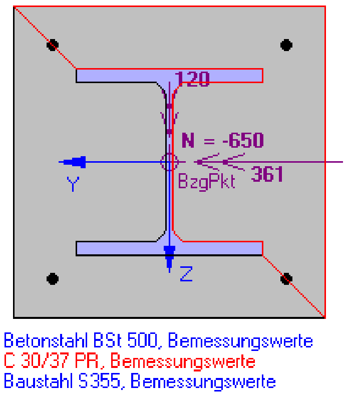
A look at the 3D stress state shows the load-bearing behavior with the different stresses. In this case, the steel stresses are not compressed, so that the concrete compressive stresses are correspondingly small.

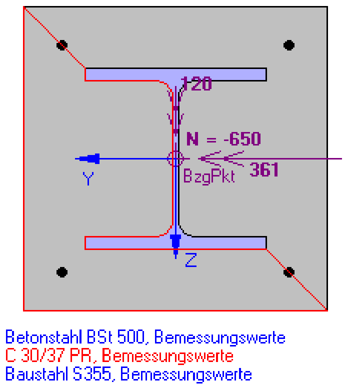


An alternative modeling without a cutout and polygons on top of each other is:

Polygon No. 1 No. 2 No. 3





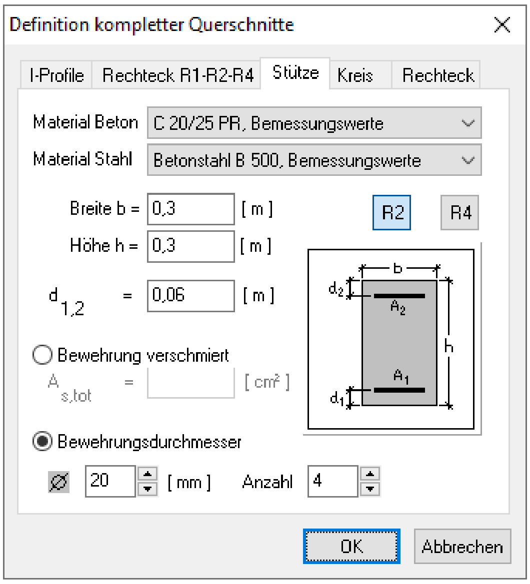


Modeling:

1. Creating a HEB 240 Beam as a Cross-Section with S355
2. Create the 4 vertices of the concrete column, but do not yet create a polygon
3. Create polygons 2 and 3 manually with the mouse by connecting the vertices and the corresponding points from the steel beam

8.5 Example 5 - Subsequently added column cross-section

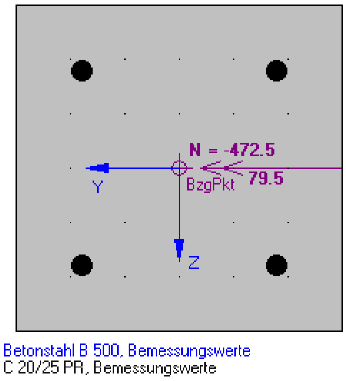
For the rehabilitation of a damaged column cross-section or the reinforcement of an existing column, the following modelling can be chosen. In the first step, the initial state before the damage to the column is modeled. The original dimensions as well as the design loads of the impacts are taken into account.



Impacts, Design

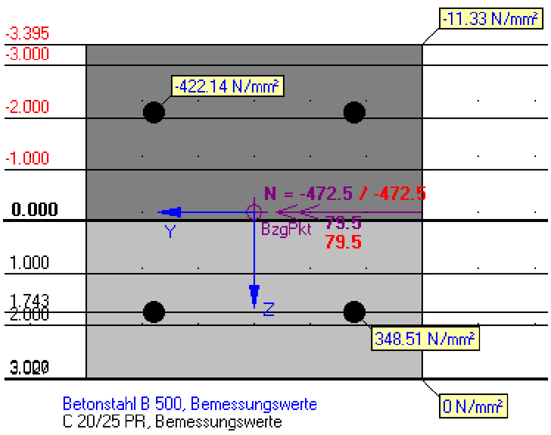
N_{Sd} = 1.35 · 150 kN + 1.5 · 180 kN = -472.5 kN

MSd = 1.35 · 20 kNm + 1.5 ·35 kNm = 79.5 kNm

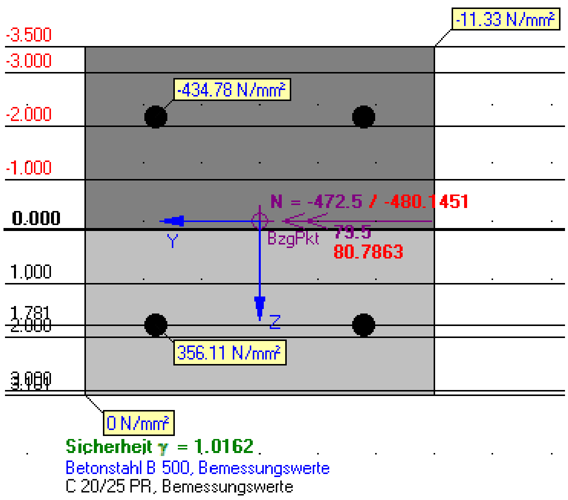


The cross-section is almost completely exploited for this impact:

Elongation Condition:

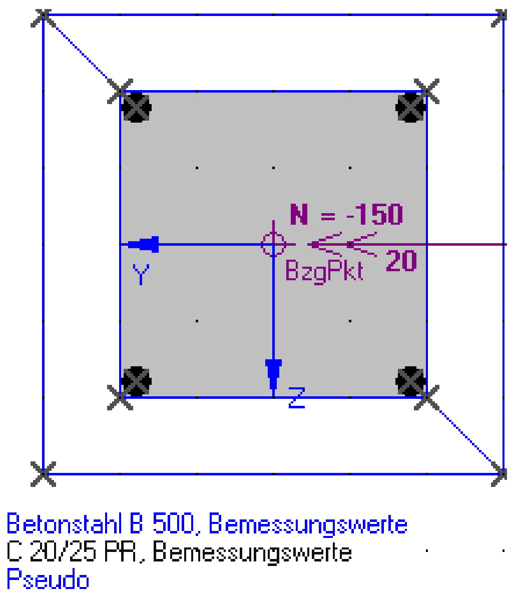


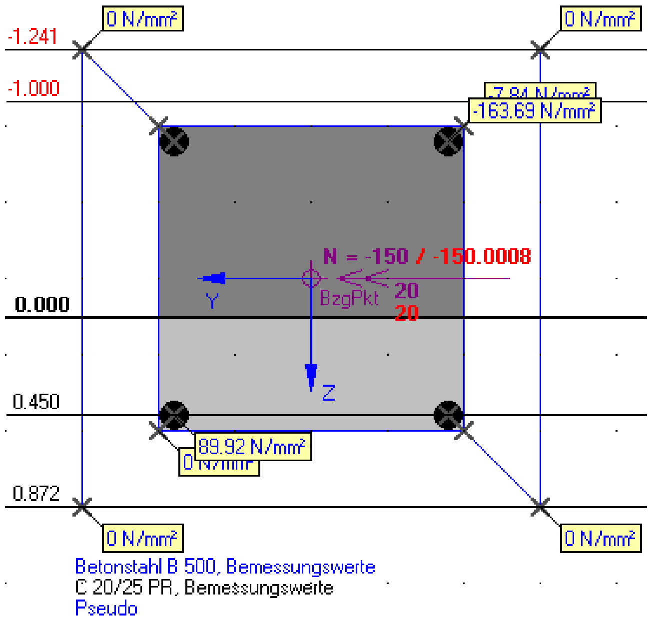
Safety Proof:



In this example, it is assumed that the concrete is damaged on all sides, in which the concrete is removed during the renovation up to approximately the stirrup reinforcement. For this purpose, the load-bearing cross-section must be reduced, for example in the chosen example by 5 cm on all sides. To visualize the spalled concrete, a "pseudo" material with Ec = 0 is defined. In order to be able to continue working with these polygons in the next steps, the following modeling with 2 individual polygons is chosen.

During the renovation, this example assumes that there is no traffic load. Only 1.0 times the dead weight, chosen in the example with N_{Sk} = 150 kN and M_{sk} = 20 kNm, remains as the actual influence, for which the subsequent elongation state results.



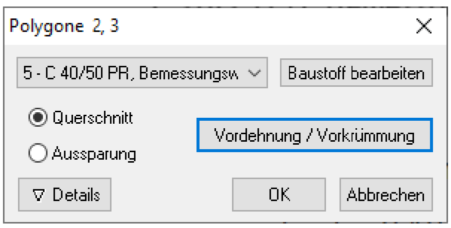


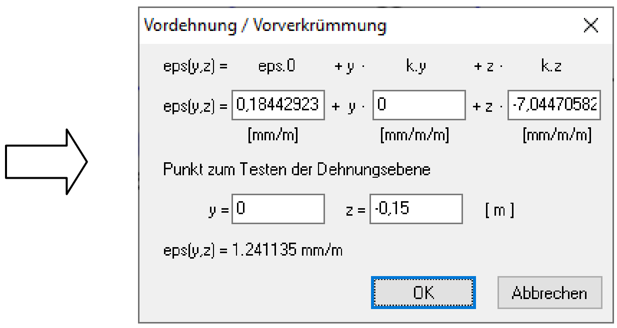
The numerical results show the following strain plane for this action with:

eps(y,z) = -0.18442923 + y * 0 + z * 7.04470582

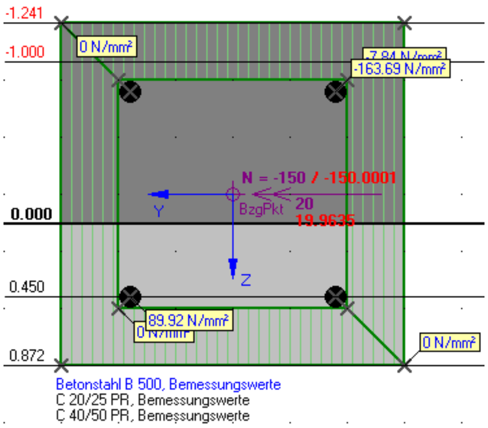
In the next step, the reduced column cross-section will be supplemented with a new concrete layer in C40/45. In any case, it must be checked whether additional ironing or additional glued-in reinforcement is required in order to permanently connect the supplemented concrete cross-section to the old cross-section. On the supplemented polygons, the previously determined strain plane with changed sign is to be applied as pre-expansion / pre-curvature.

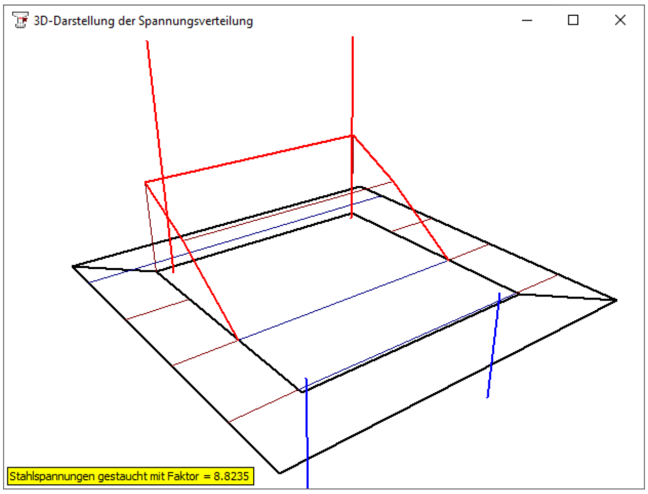
Properties of the polygons:



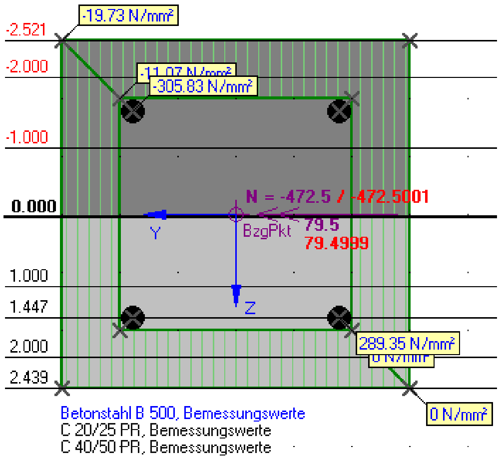


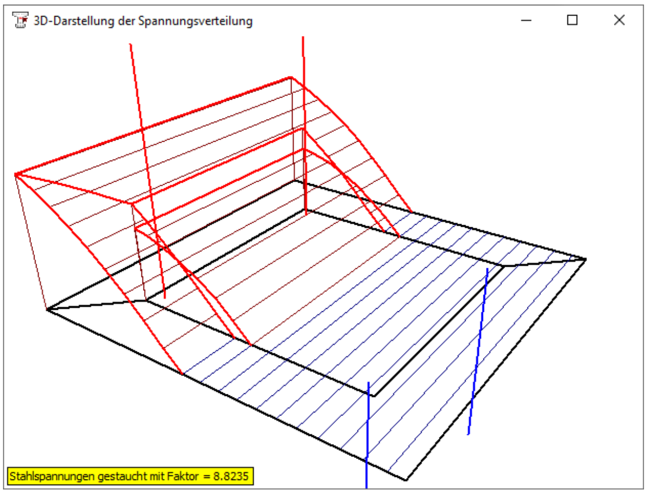
For the load condition "1.0 times dead weight", the definition of pre-elongation / pre-curvature ensures that the supplemented cross-section (the two polygons) remain stress-free for the characteristic effect in the construction state. A comparison of the stresses of the concrete and the reinforcement of the old cross-section shows exact agreement.



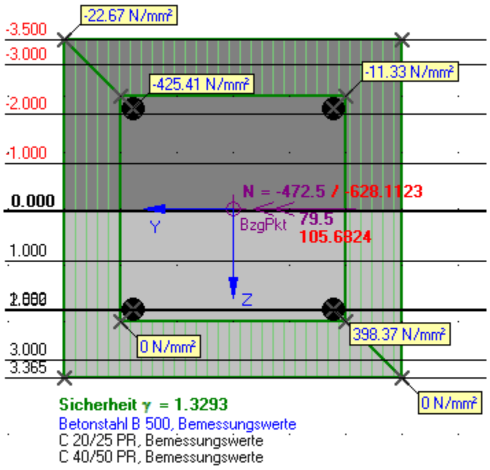


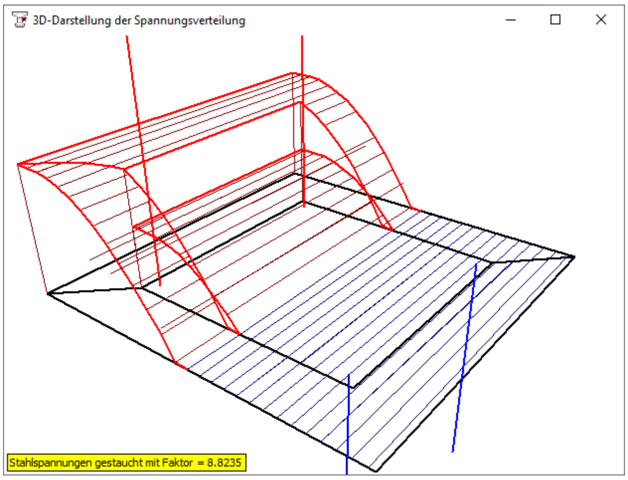
After the renovation, the traffic load will be reloaded. In addition, a sufficient cross-sectional load capacity must now again be demonstrated for the design internal forces (N_{Sd} / MSd). In the results you can see the different stresses in the different polygons.





In the present example, the choice of a higher-strength concrete for the addition increases the calculated cross-sectional load-bearing capacity.





The edge elongation of the added concrete is shown in the graph as εPανδ = -3.5 mm/m. However, the real "stress-generating" strain at the edge is only:

εEδγε = 3.5 mm/m – 1.241 mm/m = 2.259 mm/m

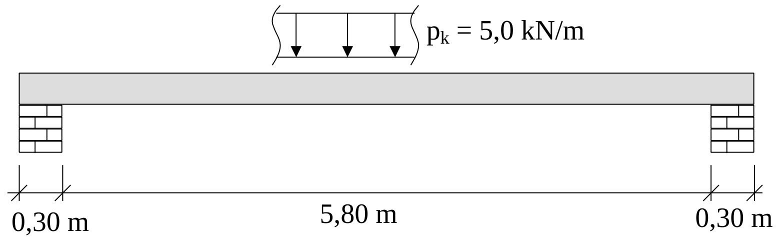
```
Polygon Nr. 2
---------------------------------------------------------------------
   Material  : C 40/50 PR, Bemessungswerte (Beton)
   Vordehnung: eps =  0.1844 + y *  0.0000 + z * -7.0447
   Rechnung im Riss zur Spannungsermittlung, f.ct = 0
   Punkte (6 Stk.) - Koordinaten (Y / Z)     eps [mm/m]  Sigma[N/mm²]
        1  ( 0)       0.1500 /     0.1500       2.4925       0.0000
        2  ( 1)      -0.1500 /     0.1500       2.4925       0.0000
        3  ( 9)      -0.1000 /     0.1000       1.7006       0.0000
        4  ( 8)       0.1000 /     0.1000       1.7006       0.0000
        5  (11)       0.1000 /    -0.1000      -1.4670     -21.0566
        6  ( 3)       0.1500 /    -0.1500      -2.2589     -22.6667
```

Analogous to the above-mentioned addition with concrete, an additional reinforcement can also be added. For this purpose, new reinforcement bars must be defined, whereby a new "reinforcement group" must be created. For this new reinforcement group, a pre-expansion + pre-curvature must also be defined so that this reinforcement remains stress-free during the effect of 1.0 x dead weight in the construction state.

8.6 Example 6 – Deformation calculation of a single-span beam

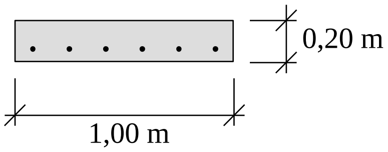
Subsequently, based on the mean values of the material properties, a deformation calculation is to be carried out on a single-span beam.

8.6.1 System



Span Leff = 0.1 + 5.80 + 0.1 = 6.00 m

Cross section



Reinforcing steel B 500

Concrete C 30/37, Ec ≈ 32,000 N/mm²

Edge distance reinforcement d1 = 4 cm

Flexural stiffness B = EI = 21333.3 kNm²

8.6.2 Load

	Limit state of load-bearing capacity	Merchantability
Dead load	1.35 · 0.2 · 1.0 · 25 = 6.75 kN/m	0.2 · 1.0 · 25 = 5.0 kN/m
Live load	1.5 · 5.0 = 7.50 kN/m	30% · 5.0 = 1.5 kN/m (quasi-constant) 50% · 5.0 = 2.5 kN/m (common) 70% · 5.0 = 3.5 kN/m (rare)

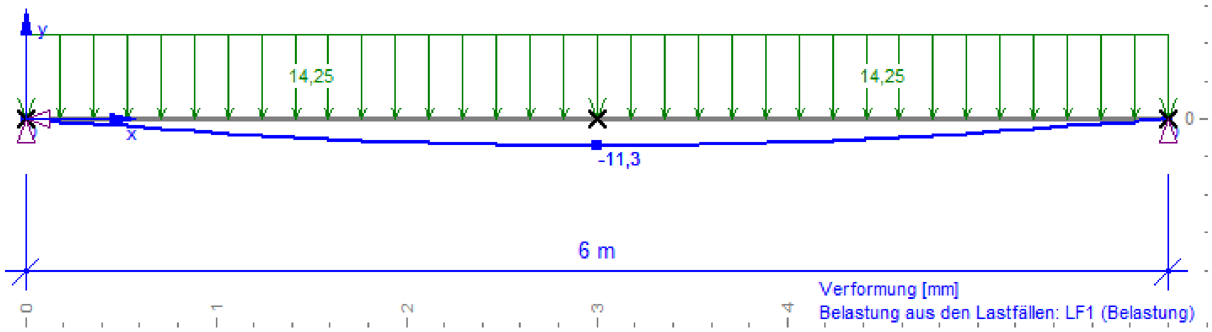
8.6.3 Design in the Load-Bearing Capacity Limit State

Bending moment $$M = \frac{q \cdot \lambda^2}{8} = 64{,}125 \text{ kNm}$$

In order to determine the reinforcement, the design values of the material properties are required. Model the cross-section with the dimensions given above. When choosing the reinforcement, you first specify any surface. Then carry out the ***design***
(Results menu).

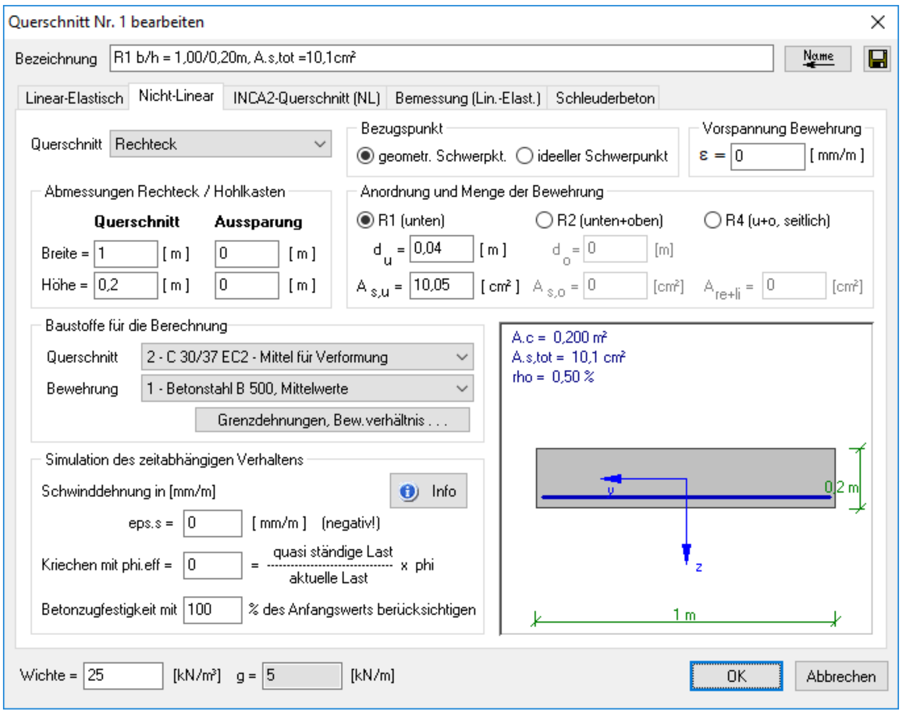
Reinforcement req A_s = 10.05 cm² / m

Deflection $$w = \frac{5}{384} \cdot \frac{q \cdot \lambda^4}{EI} = 0{,}0113 \text{ m}$$ (linear-elastic)



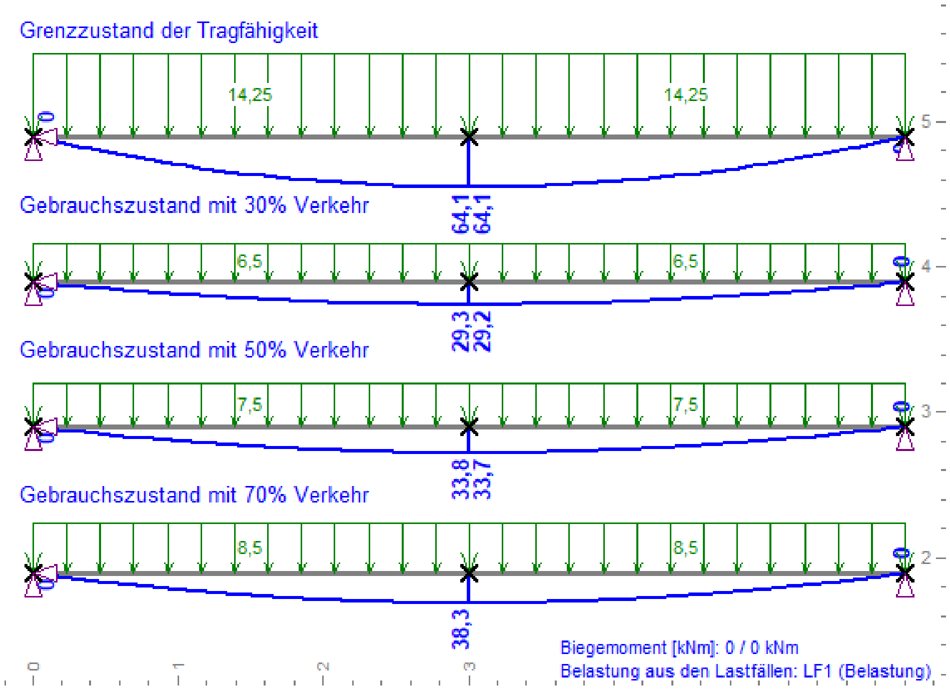
8.6.4 Deformation in the state of service

The system is modeled in the Stab2D-NL program. To calculate the deformations in the service state, the mean values of the material properties must be used. These can be imported from the materials table.

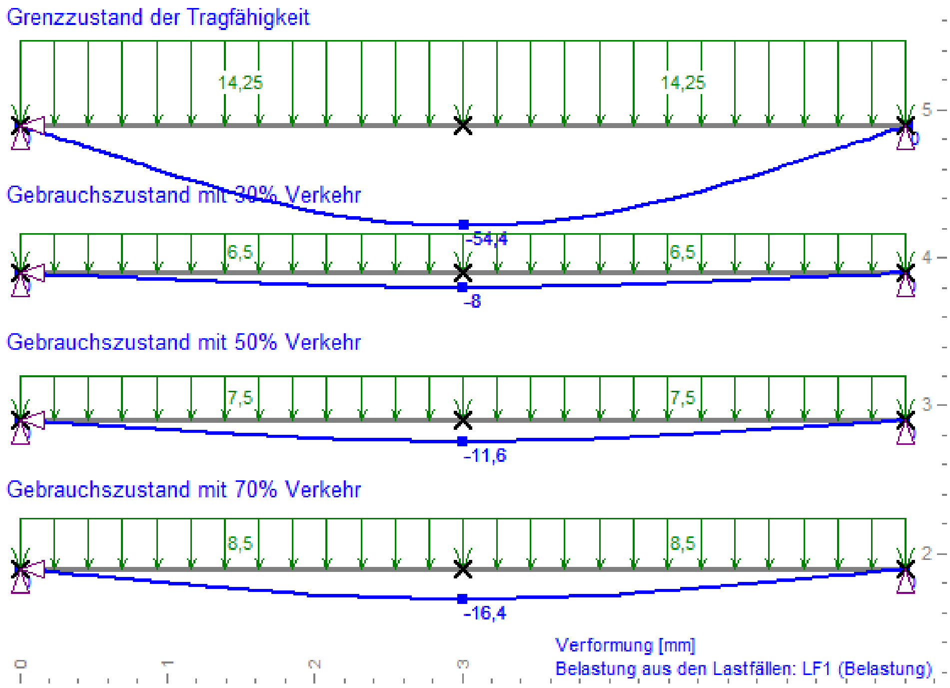


The nonlinear calculation provides the following results in graphical form.

Bending moment:



Deformation, vertical in [mm]



Deformation in the state of service

30% traffic loadw = 8.0 mm

50 % traffic loadw = 11.6 mm

70% traffic loadw = 16.4 mm

Limit state of load-bearing capacity w = 54.4 mm

It should be noted that these values are obtained at t = 0. Taking into account shrinkage and creep, the deformations will continue to increase. Furthermore, it should be noted that the conditions on the construction site and the resulting material properties are difficult to determine. The deformations calculated in this example do not represent an absolute measure, but only show the range in which the deformations are to be expected.

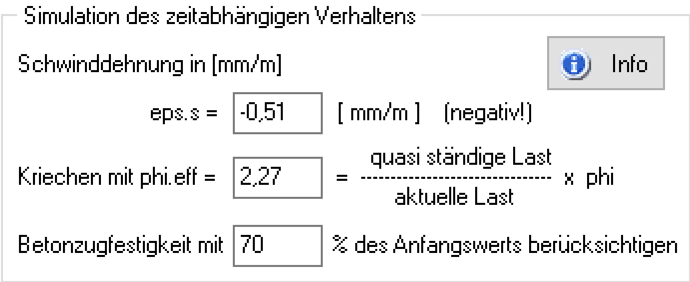
8.6.5 Deformation in the state of use with shrinkage and creep

According to the formulas from EC2 (2000 edition), the following creep and shrinkage values were determined. The humidity was assumed to be 60%.

Wane $\varepsilon\sigma$ = -0.51 mm/m

Creep φ = 2.27

Adjustment of cross-section for quasi-permanent loads for t = ∞



Within the program (calculation module of INCA2) the shrinkage of the concrete is modeled as a pre-extension of the reinforcement. Creep is taken into account within the program by stretching the stress expansion lines of the concrete by the factor (1 + φ). As a result, the values for the Concrete Materials Act change as follows:

Modulus of elasticityE = 31,344 / (1+2.27) = 9,585 N/mm²

Strain $\varepsilon\chi 1$ = - 2.2 · (1+2.27) = - 7.194 mm/m

Strain ε_{Cr} = 0.11 · (1+2.27) = 0.3597 mm/m

The Participation Act for the Concrete Tensile Zone will also be amended within the programme as follows:

Strain ε_{Cr} = 0.11 · (1+2.27) = 0.3597 mm/m

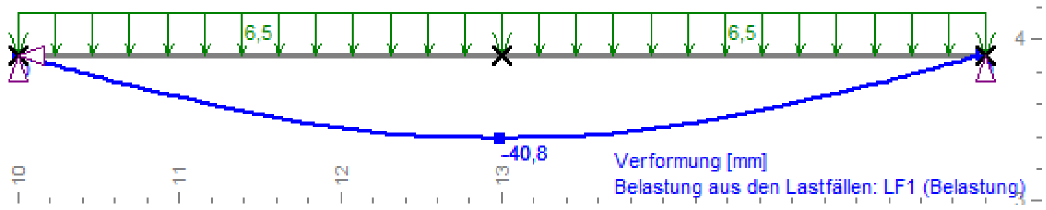
Analogously, the limit strains of the concrete have to be adjusted:

$\varepsilon\Pi\rho\varepsilon\sigma\sigma\upsilon\rho\varepsilon$ = - 3.2 · (1+2.27) = 10.464 mm/m

$\varepsilon_{Druck.z}$ = -2.2 · (1+2.27) = 7.194 mm/m

In addition, a reduction of the stiffening contribution of the concrete tensile zone for t = ∞ to 70% is modelled, which is a good empirical value.

With these changes, the state for the quasi-constant load is calculated. For the other load situations (e.g. "rare" or limit state of the load-bearing capacity) the value for $\varphi_{eff\ would\ have\ to}$ be adjusted, since proportionately only the quasi-permanent loads have a creeping effect.

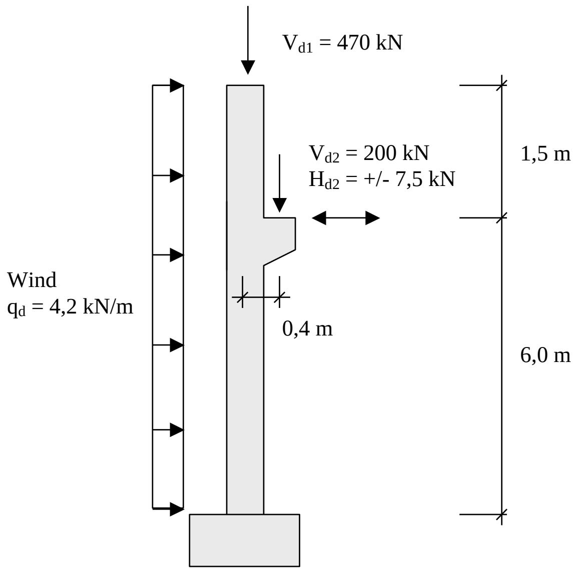


For more information on creep and shrinkage modeling, see chapter 6.12 Shrinkage and creep.

8.7 Nonlinear column calculation with INCA2 and ABaS

Using the example of a cantilever column (prefabricated construction, industrial hall), various calculation processes are to be demonstrated.

ABaS (Illustrative Beam Statics) was also developed in the Concrete Structures Department of the Technical University of Hamburg-Harburg and can be downloaded free of charge from the institute's homepage (www.mb.tu-harburg.de).



Rectangular cross-section:

w / h / d1 /d2= 40 / 45 / 5 / 5 cm

Basically, the design of a slender column by means of a non-linear calculation is always iterative, i.e. the structural safety verification can be provided for a selected reinforcement or not. To simplify this iteration in this example, a calculation using the simplified model support method is performed in advance. This allows a reinforcement to be determined, which in turn is the initial value for a nonlinear calculation with INCA2 and ABaS.

To simplify matters, combination coefficients are not taken into account.

8.7.1 Manual calculation with the simplified model support method

Bending moment according to first-order theory, including unintentional centering (misalignment)

$$M_{d,1} = 4{,}2\ \text{kN/m} \cdot \frac{7{,}5^2}{2} + 7{,}5\ \text{kN} \cdot 6{,}0\ \text{m} + 200\ \text{kN} \cdot 0{,}4\ \text{m} + \frac{1}{200} \cdot \left(6{,}0\ \text{m} \cdot 200\ \text{kN} + 7{,}5\ \text{m} \cdot 470\ \text{kN}\right)$$
$$= 266{,}75\ \text{kNm}$$

Centering and bending moment according to second-order theory

L0 = 2 · L

at x = 7.5 m $\quad e_2 = \frac{L_0^2}{2070 \cdot d} = \frac{(2 \cdot 7{,}5\ \text{m})^2}{2070 \cdot 0{,}40\ \text{m}} = 0{,}2717\ \text{m}$

at x = 6.0 m $\quad e_2 = \frac{(2 \cdot 6{,}0\ \text{m})^2}{2070 \cdot 0{,}40\ \text{m}} = 0{,}1739\ \text{m}$

Md,2= 0.2717 m · 470 kN + 0.1739 m · 200 kN = 127.70 + 34.78 = 162.48 kNm

Consideration of the shape of the moment curve:

Wind stress and horizontal force (braking force crane) result in a triangular bending moment curve. The bending moment can therefore be reduced as follows:

$$M_{d,2} = M \cdot \frac{10}{12} = 135{,}40\ \text{kNm}$$

The design internal forces are therefore as follows

MSd = 266.75 + 135.40 = 402.15 kNm

NSd = 470 kN + 200 kN = - 670 kN

A design with INCA2 provides a required reinforcement of $A_{s,tot}$ = 37.66 cm²

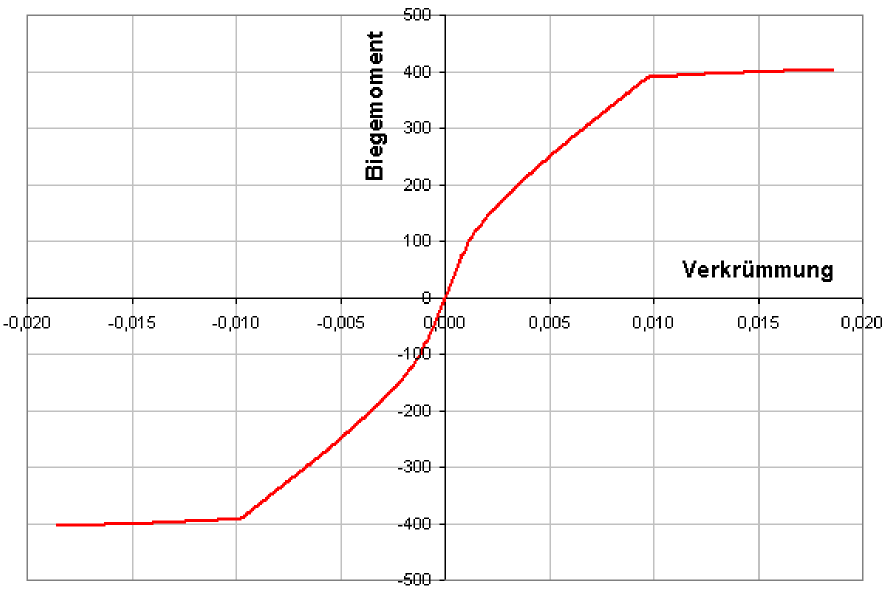
8.7.2 Nonlinear computation with INCA2 and ABaS

In principle, the deformations are also calculated here on the basis of the mean values of the material properties in order to obtain a realistic behavior of the column. However, since the resulting internal forces depend to a very large extent on the properties of the material (poor concrete⇨, more deformation, ⇨ greater stress), the material properties for such structures are already subject to the partial safety coefficients for the determination of internal forces. The concrete strength is therefore $f_c = f_{cm} / 1.5$, the steel strength $f_y = 500 / 1.15 = 435$ N/mm².

The background to this approach is that in the new safety concept according to EC2 or DIN 1045-1, the relevant scattering variables should always be subject to a partial safety coefficient.

A "B 500, design values" is chosen as the material for the reinforcement ($f_{yd} = 435$ N/mm²), and "C30/37, column design" is chosen for the concrete ($f_c = 38 / 1.5 = 25.333$ N/mm²). From this cross-section, the Mk line is calculated for a normal force of N = -670 kN. The change in the normal force in the range from 6.0 m to 7.5 m with N = -470 kN is neglected because the influence of the column tip on the overall result is very small.

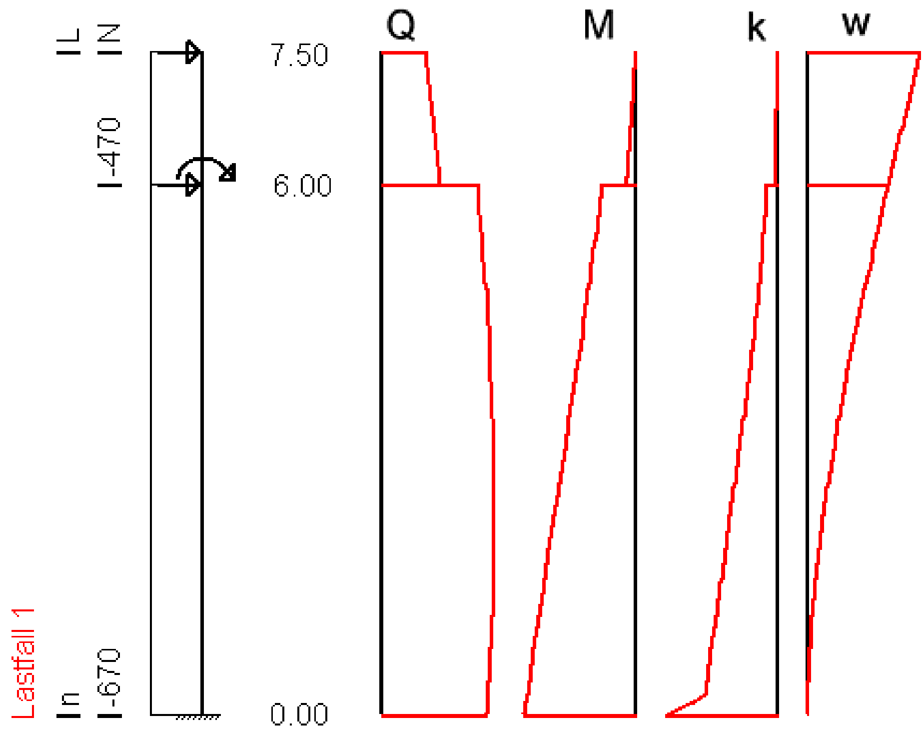
M/k-line for N_{Sd} = -670 kN and $A_{s,tot}$ = 37.66 cm²



In the following, a nonlinear calculation is performed with this M/k line (here with the program ABaS), which yields the following results.

MSd = -354.73 kNm NSd = -670 kN eHead = 0.1434 m

As can be seen, the achieved bending moment of M_{Sd} = 354.73 kNm is lower than the maximum absorbable of M_{Rd} = 402.15 kNm. In an iterative calculation, one can reduce the reinforcement from $A_{s,tot}$ = 37.66 cm² to $A_{s,tot}$ = 34.0 cm², the absorbable moment is then reduced to M_{Rd} = 374.3446 kNm. An ABaS calculation with a Mk line for the changed cross-section results in an acting bending moment of M_{Sd} = 372.58 kNm at the clamping with a head deformation of w = 17.24 cm. However, a further reduction of the reinforcement is no longer possible because the cross-section tears open too far (already plasticized at the clamping point, see graphic), the stiffness decreases sharply and the deformations and thus the effects increase faster than the resistance of the cross-section in the event of increasing curvatures.



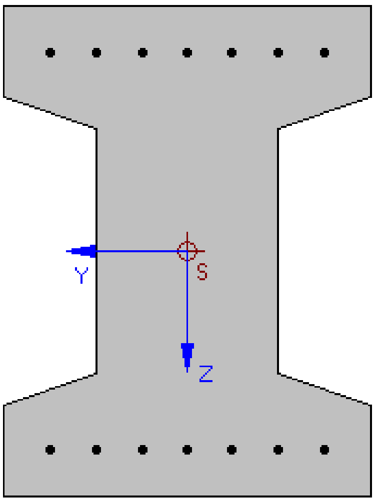
The computer calculation shown is very complex due to its iterative nature, but delivers the exact results within the framework of the assumed material parameters. In addition, it is only through the nonlinear calculation that the existing moment and curvature curve can be recognized and a different factor can be selected accordingly for the model support method.

8.8 Interaction Diagrams N_x / My

In the case of repetitive calculations of the reinforcement for identical or scaled cross-sections, it may be useful to create an M/N interaction diagram to simplify the work. In particular, such a diagram may be necessary if it is a cross-sectional type not tabulated in the literature or if another reinforcement with yield strength different from the B 500 is used. With INCA2, you have the ability to create both a dimensioned and a dimensionless interaction diagram.

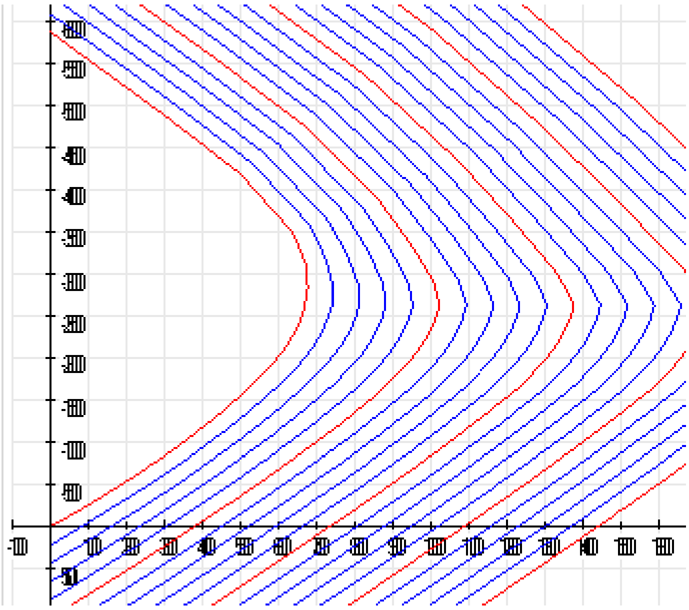
8.8.1 Dimensioned interaction diagram

For the cross-section shown below, an interaction diagram will be created. In the supplied sample datasets, you will find the INCA2 file Prop-Double-T.inc. The reinforcement area of all reinforcing bars was chosen in the sum of $A_{s,tot}$ = 5.0 cm².



Now select the calculation of the M/N line in the Results menu. In the window that opens, select Create multiple ***lines***. Continue to enter in the text boxes that the reinforcement should vary by a factor of 0 to 20 and that a total of 21 lines should be calculated.

After clicking on the ***OK*** button , it may take a little moment, depending on the computing power of the computer, until the numerical and graphical results are displayed. With the settings shown, you get an M/N diagram in which the individual lines differ by the amount of reinforcement of 5 cm². The line on the far left was calculated as As = 0 cm², the line on the right side as As = 100 cm².



For better identification, every fifth line is displayed with a different line width and color. Line colors and line thicknesses can be selected in the *menu Tools* => *Settings* => *Image Material Law* with lines 1 and 2.

8.8.2 Dimensionless interaction diagram

Dimensionless diagrams are used to design similar cross-sections. If we take a closer look at the input values μ and ν , we can see that this results in a cross-section with the dimensions b = h = 1.0 m, which has the concrete strength f_c = 1.0 MN/m². The value to be read ω represents the mechanical reinforcement ratio, i.e. the ratio between the maximum compressive force of the reinforcement and the concrete.

In the following, the calculation is carried out on the basis of DIN 1045-1 (file Einheitsquerschnitt.inc).

For the range of large normal forces, it must be taken into account that, in derogation from Table 9 (DIN 1045-1), the maximum compressive elongation may be increased from ε = - 2.0 mm/m to ε = - 2.2 mm/m for cross-sections pressed centrally or slightly off-center without risk of kinking. This means that the reinforcement can also be fully exploited in this area (according to Betonkalender 1/2002, p. 314, point 4.1.5).

Based on the definition of the mechanical reinforcement ratio, the following value is to be used as reinforcement:

for ω = 1.0: Fc = F_s

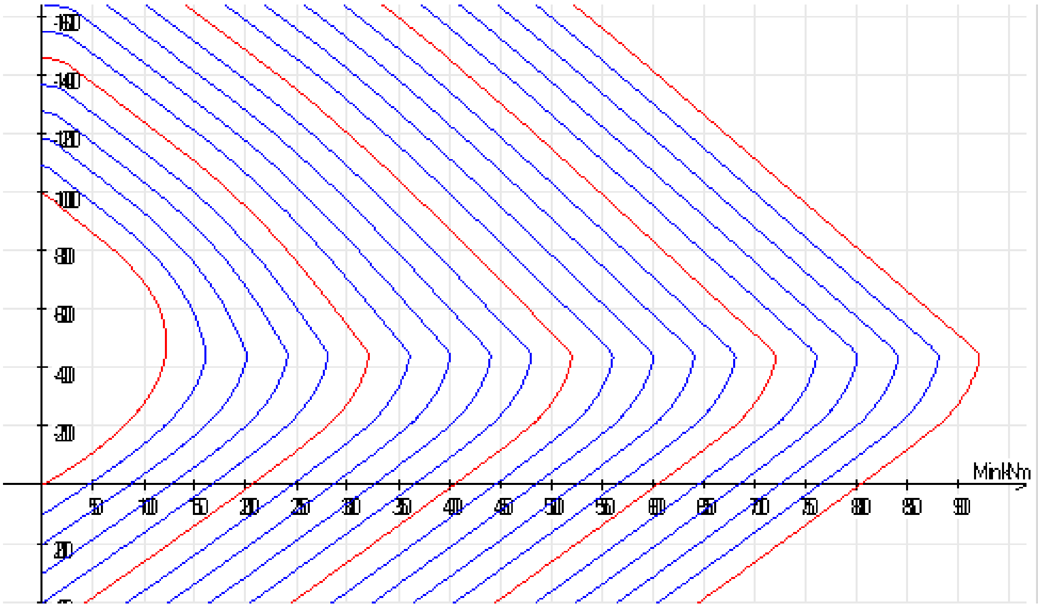
1.0 MN/m² = As · 434.78 MN/m²

As = 23.000138 cm²

When calculating the MN lines, the following settings must be made with this selected reinforcement:

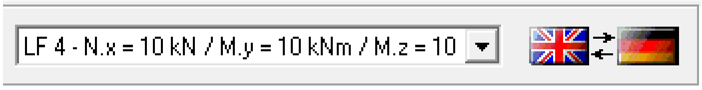
- Reinforcement vary by factor from 0.0 to 2.0
- Number of lines: 21

This results in the following diagram:



9 Language

For the interface of INCA2 you can switch between German and English with the button in the upper right corner. Results etc. will then also be displayed in the corresponding language.



This help file is only available in German.

10Authors

10.1 Development

The program MasQue as a precursor of INCA2 for the calculation of arbitrary solid cross-sections under biaxial bending with longitudinal force was developed from 1987 onwards at the Technical University of Hamburg-Harburg in the field of concrete construction (3-07). The basic calculation algorithms were developed by Prof. Dr.-Ing. Ulrich Quast in the early 1970s. Extensions were made by Dipl.-Ing. Dirk Busjaeger. Dr.-Ing. Marek Los adapted the MasQueW program to the Windows interface as part of his work at the TUHH.

The programming of the interactive graphical user interface as the program INCA2 was carried out in 1998/99 by Dr.-Ing. Uwe Pfeiffer with Delphi 4.0 ™ and subsequent versions. Initially, the computational core of MasQueW was used. However, since there was a constant further development up to today's program scope, the reprogramming of the calculation algorithms was also necessary from 2000 onwards. This was particularly true of the new material laws in order to be able to better map the deformation behavior.

The overarching goal was to create a powerful tool for the non-linear investigation of planar trusses in order to investigate structures with constrained loads, multi-part, coupled columns or reinforced concrete structures in general in their entirety (interaction between column and beam). INCA2 was an important building block for this in addition to the Stab2D-NL framework program. On the one hand, existing, mostly commercial programs were rarely able to calculate non-linearly, and on the other hand, some correlations (e.g. stiffening of the cracked concrete tensile zone) were formulated differently than desired by the author. Since the commercial programs are usually only designed for the design of reinforced concrete structures, but in our own work systems with predefined reinforcement should be loaded up to the limit state, the next reason for the development of a separate program arose. All in all, the limited possibilities for intervening in the calculation method of commercial programs are advantageous for daily practice in the engineering office, but are usually not sufficient for scientific investigations.

Since 2002, the INCA2 and Stab2D-NL programmes have been successfully used in diploma and advanced theses in various fields (solid construction, steel construction, masonry construction, research into new construction methods). Since 2003, the programs have been used in a number of engineering firms, construction companies and at a large number of universities to solve rather unusual problems. The use of the program in engineering offices in practical use resulted in a number of improvements, especially to the user interface of the program. Suggestions are therefore welcome. New versions of the program can be found on the Internet at approximately half-yearly intervals at

www.u-pfeiffer.de.

Further literature on the calculation algorithms for the program MasQue (predecessor of INCA2) can be obtained at cost price from the Institute of Concrete Structures of the TUHH (Issue 415 DAfStb, Beuth Verlag, Berlin, 1990) – at least as long as stocks last.

Title: Program-controlled calculation of arbitrary solid cross-sections under biaxial bending with longitudinal force (MasQue program)

Authors: Dipl.-Ing. Dirk Busjaeger, Univ.-Prof. Dr.-Ing. Ulrich Quast

Year: 1990

10.2 Form of distribution of the program / licensing

The program INCA 2 is freeware, which means that it may be copied and passed on in the context of non-commercial use at universities, by students or employees, but also in commercially working engineering offices, etc. There is no license fee to be paid to the author.

However, commercial users as well as universities etc. in particular are asked to register. This gives me an overview of where and for what purposes the INCA2 programme is being used and where there may be a need for adjustments to the programme.

To register, please go to the following homepage:

www.u-pfeiffer.de

and select INCA2 and then Licensing. Please fill out the form and submit it at the push of a button. After a short processing time, a licensing file will be sent to you by e-mail, which you can copy into the program directory of INCA2 (or replace the old file). There are two headers stored in this file, which are always output when printing from INCA2. Therefore, do not forget to enter the name and address of the institute or engineering office in these lines in the form on the Internet.

Updates to the INCA2 programme are made at irregular intervals. In the process, errors are eliminated and/or suggestions from the users are incorporated. In the event of minor changes, the new version will be made available for download on the Internet without further notice. In the event of major changes, an information letter will be sent by e-mail to the registered users.

Therefore, please check the above homepage from time to time to be able to use the latest version of INCA2. The licensing file remains valid for new versions of the program.

10.3 Disclaimer

Although the program has been developed and tested to the best of our knowledge, it cannot be guaranteed to be completely error-free. Therefore, it is expressly pointed out at this point that no liability can be assumed for any damage caused by the use of the program.

In this context, it should be pointed out that every computer calculation should be checked by a responsible engineer in a rough or logical way!

Hints and suggestions for improvement are always welcome. In particular, we are interested in whether this program is suitable for teaching purposes and how it can be used appropriately.

If you have any questions or suggestions, please write to:

Dr.-Ing. Uwe Pfeiffer

email: pfeiffer@tuhh.de

http://www.u-pfeiffer.de

You can always find the current postal address as well as the availability by phone and e-mail on the Internet at www.u-pfeiffer.de.

10.4 Author

Dr.-Ing. Uwe Pfeiffer

Birthday	1974
1992	Abitur, Goethe-Gymnasium in Schwerin
1994 - 1999	Studied civil engineering and environmental engineering at the Technical University of Hamburg-Harburg, Specializations: Concrete Structures, Steel Construction, Construction Mechanics/Structural Analysis
1999 - 2004	Assistant in the Department of Concrete Structures (TUHH) under the direction of Univ.-Prof. Dr.-Ing. U. Quast
2004	PhD thesis on the topic: The non-linear calculation of 3D-frames made of reinforced or pre-stressed concrete with consideration of the axial elongation caused by cracking
since 2004	Employee at the Consultant Sellhorn, Hamburg, www.sellhorn-hamburg.de Head of Structural Engineering, Authorized Director, Projects especially in hydraulic steel construction, quay walls national and international, jointless structures, repair of historic buildings, bridges

Developed Programs

since 1993	***Vector Ace*** - Mathematics program for the high school level on vector calculus (programming language Turbo Pascal 6.0, MS-DOS)
since1997	***Truss2D-NL*** - Program for the nonlinear calculation of trusses, taking into account large deformations and nonlinear material laws with permanent deformation (programming language Visual Basic, from Win 3.11)
since 1998	***INCA2*** - Program for the interactive calculation of solid cross-sections under biaxial bending with normal force (programming language Delphi 4.0, from Win 95)
since 2001	Development of ***Stab2D-NL*** (Programming language Delphi 4.0 and following)

formerly Institute of Concrete Structures, Technical University of Hamburg-Harburg

last change: 2024-03-05