

IDEA StatiCa Detail

Theoretical background



October 2018

Content

1 Geometric model.....	4
1.1 Geometry of concrete parts.....	4
1.2 Opening and the bent end of the beam.....	6
1.3 Bearings.....	7
1.3.1 Point distributed.....	7
1.3.2 Point bearing plate.....	7
1.3.3 Line support.....	8
1.3.4 Hanging support.....	8
1.3.5 Patch support.....	8
1.4 Load transfer elements.....	8
1.4.1 Hanging load.....	9
1.4.2 Patch load.....	9
1.4.3 Bearing plate.....	10
2 Load.....	10
3 Reinforcement.....	11
3.1 Possibilities of reinforcement.....	13
3.1.1 Wall reinforcement.....	13
3.1.2 Reinforcement of beams.....	16
3.2 Finishing of reinforcement.....	16
3.2.1 Finishing of longitudinal reinforcement for EC2.....	17
3.2.2 Finishing of longitudinal reinforcement for ACI.....	18
3.2.3 Finishing of stirrups.....	18
3.3 Topological optimization.....	20
3.3.1 Analysis.....	20
3.4 Linear analysis.....	21
3.5 Optimization of reinforcement of areas.....	21
4 Analysis.....	22
4.1 Physical model.....	22
4.1.1 Physical model – ultimate limit state.....	23
4.1.2 Physical model – serviceability limit state.....	24
4.2 Mechanical model.....	24
4.3 Analysis.....	25
5 Checks.....	25
5.1 Checks according to EC.....	25

5.2 Checks according to ACI 26

6 Literature 27

1 Geometric model

With IDEA StatiCa Detail application, models can be developed to solve discontinuity problems occurring in concrete structures. Typically, the model consists of concrete parts, support, load transfer elements and reinforcement. The computational model is limited to planar tension, both in the case of wall analysis and the analysis of detail of 1D elements (beams and columns).

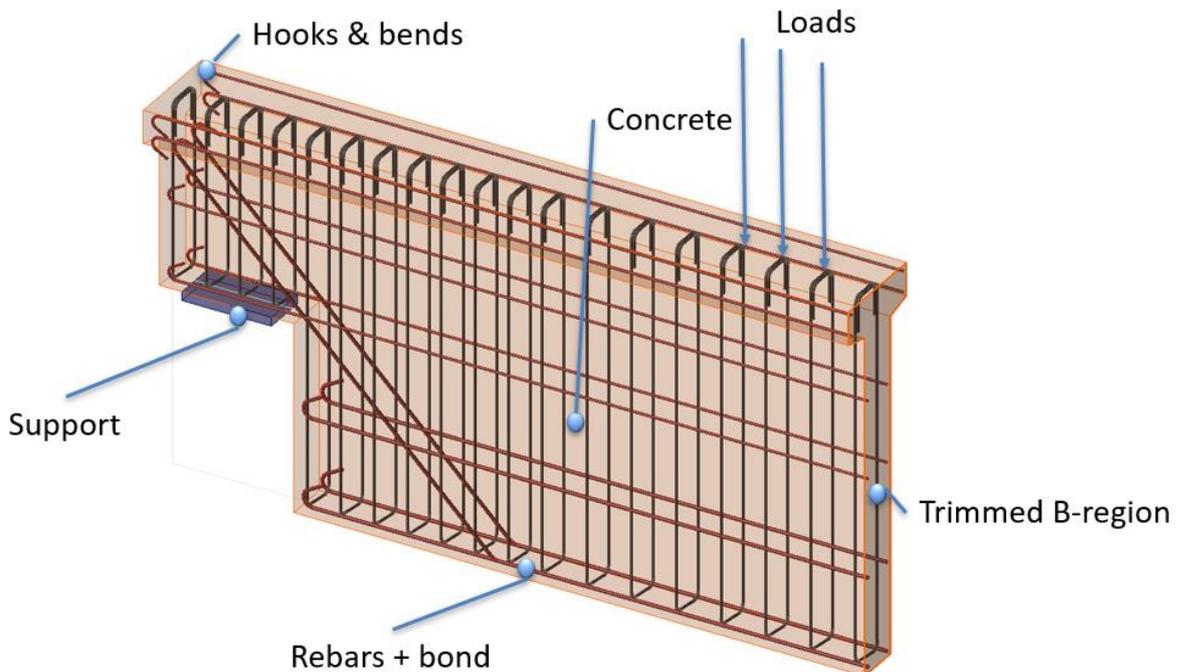


Fig. Computational model of the structure part

1.1 Geometry of concrete parts

The concrete parts of the structure are modelled with isoparametric 2D wall elements (CQUAD4 or CTRIA3) and can be input using walls and 1D bars.

Walls can be input of several types. For simple input, a rectangle type was created (input by width and height of the wall). In general, the wall can be input using a polygon.

In the case of 1D bar, the geometry is input by the cross section and the 1D bar geometry. The geometry of both the cross section and the element may include haunches. The haunch cross section is replaced by a wall model with a constant (average) wall thickness. The thicknesses of each wall model are further adjusted according to the thickness of the thinner wall. The maximum thickness of each wall model is equal to the thickness of the adjacent wall + its double height. 45 ° angle of rotation is assumed.

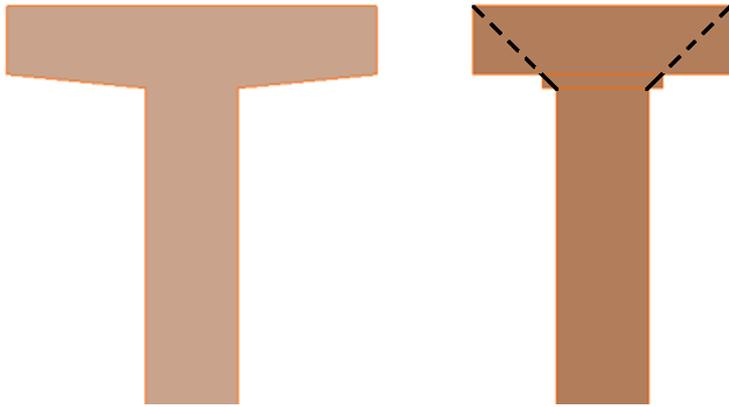


Fig. Principle of thickness determination

In the case of haunches in the horizontal plane, the bar is divided into five sections along the length. Section geometry and wall thickness are determined separately based on the initial and final sections of each section separately. For this type of haunch, the average wall thickness on the given section is calculated.

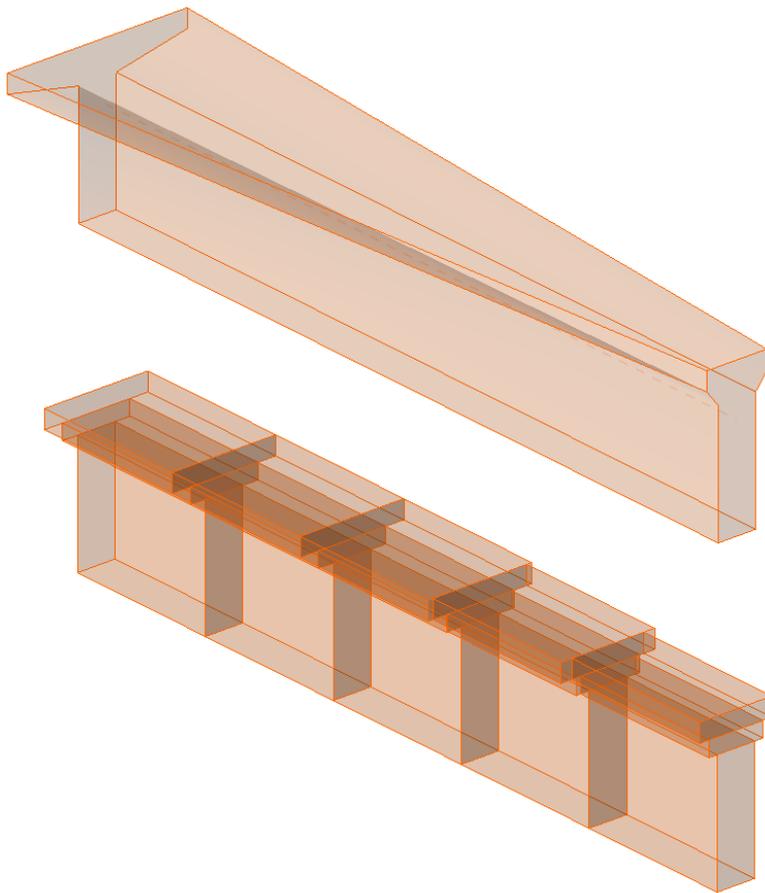


Fig. Horizontal haunch (Top Chord) - 3D vs. 2D model

The 1D bar can also be modelled as unfinished (trimmed), i.e. replacing other parts of the construction by the inner force transmission area (B area, Saint-Venant area) and internal

forces at the cutting point. The length of the Saint-Venant area is automatically determined as half of the cross section height at the cut point. Internal forces at the cutting point are automatically counted from boundary conditions and the load. The load by internal forces is applied to the control node on which the wall nodes are dependent.

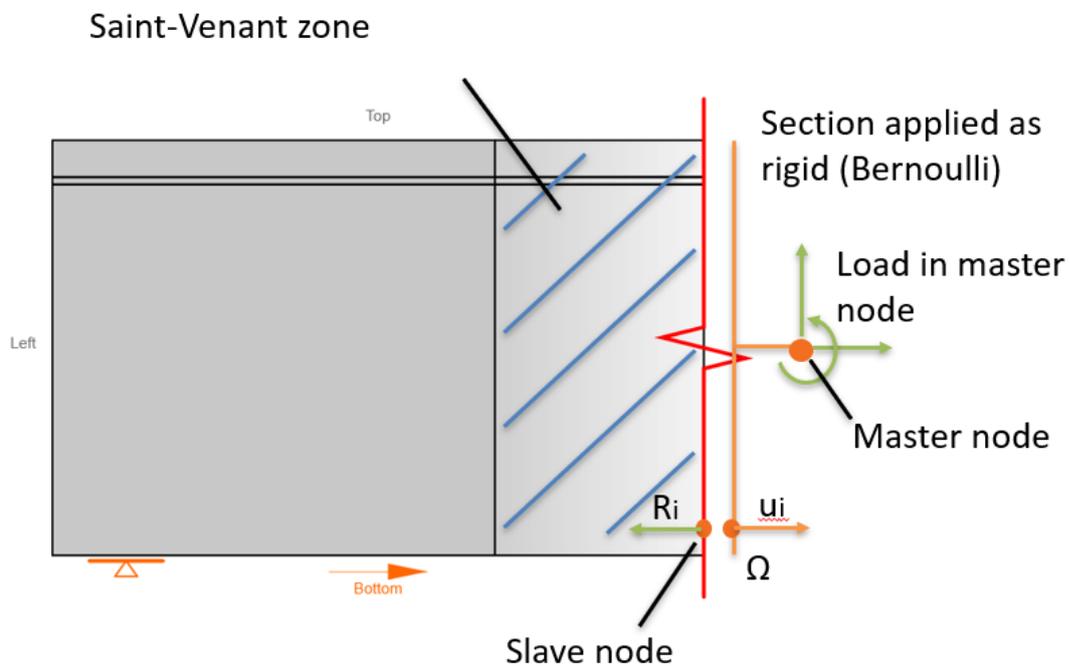


Fig. The transmission of internal forces at the cutting point

1.2 Opening and the bent end of the beam

The wall and 1D bar geometries can be modified using openings, or the bent end of the beam. The shape of the opening may be:

- Rectangular
- Circular
- Polygon

A special type of the opening is the bent end of the beam.

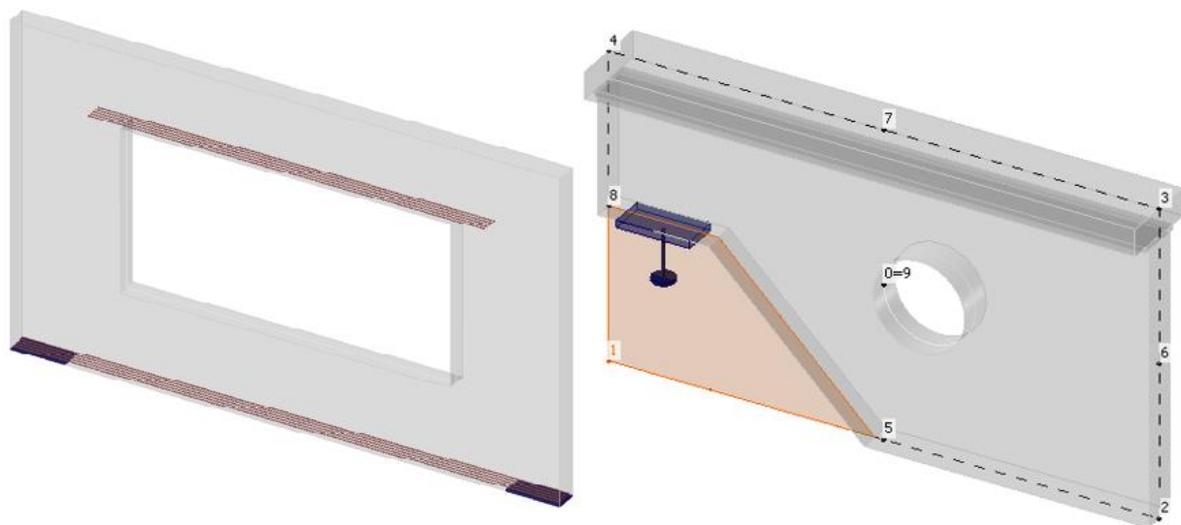
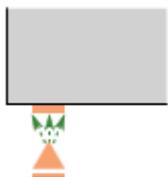


Fig. A rectangular opening in the wall and the bent end of the beam

1.3 Bearings

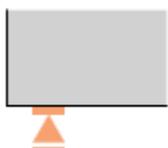
The following bearing types can be input in the IDEA StatiCa Detail for correct support of the structure.

1.3.1 Point distributed



Bearing can be placed on the edge or on the model area. By the length of the bearing or the effective radius the length / area on which the bearing will be distributed is determined. The stress is not concentrated at one point, it is distributed over a larger area and there are no rapid changes in the stress.

1.3.2 Point bearing plate



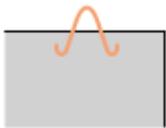
The point reaction is transferred to the model via a metal plate. The plate is not analysed, it serves only for the reaction transfer.

1.3.3 Line support



Line bearing can be input either on the edge of the structure or on the model area with any curve. Bearing can be rigid or flexible with a user-defined rigidity value. In addition, it can be input as linear or compressed only.

1.3.4 Hanging support



The bearing applied to the outside of the hanging is converted, according to the rotation, to the bearings acting in the axes of each hanging branch. These bearings are applied at the point where the hanging branches enter the concrete. The part of the hanging protruding from the concrete is not a part of the calculation model and is not analysed. The hanging can be a lifting eye, nail or pin, or a reinforcement welded to a steel plate.

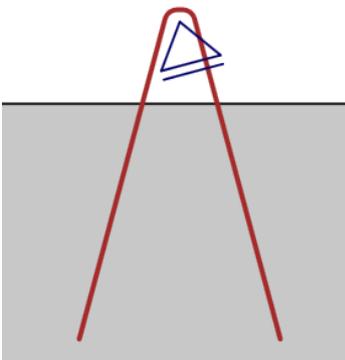


Fig. Hanging with a swivel bearing

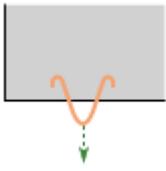
1.3.5 Patch support



1.4 Load transfer elements

IDEA StatiCa Detail offers several elements for load transfer into the construction.

1.4.1 Hanging load



The load applied to the outside part of the hanging is converted to the axial forces affecting two hanging branches. The hanging branches are difficult by the converted forces at the point where they enter the concrete. The outside part of the hanging is not part of the calculation model and is not analysed. The hanging can be a lifting eye, nail or pin, or a reinforcement welded to a steel plate.

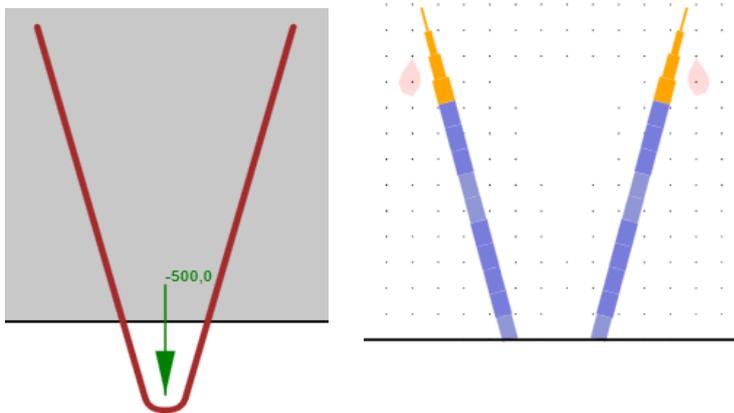
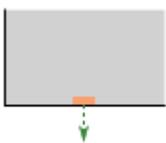


Fig. Input of the force on the hanging Fig. Stress on hanging branches

1.4.2 Patch load



Planar load is the area where the point load is input. The load is transferred to the calculation model via the reinforcement that lies in the area. This reinforcement must be explicitly specified. A planar load can be, for example, a steel plate welded to the reinforcement or a cross secondary beam affecting the primary beam. If no reinforcement that belongs to planar load is specified, the load is delivered directly into the concrete.

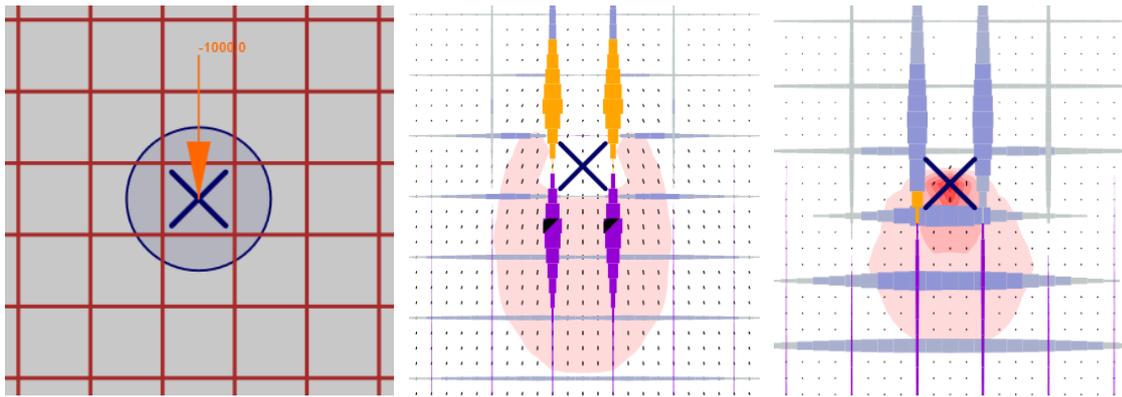
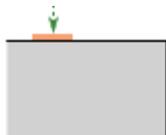


Fig. Specifying of the planar load Fig. Transfer result to reinforcement Fig. Transfer result to concrete

1.4.3 Bearing plate



The point load is spread over concrete through a metal plate. The board is not analysed, it only serves for load distribution.

2 Load

The geometric model can be loaded on the wall surface by

- Point load - load impulse at the point on the structure. This load must be correctly transferred to the structure, see the section [Load transfer elements](#)
- Line load - load impulse on the construction line
- Surface load - load impulse on wall surface
- Internal forces of the cut - the loading impulse of the cut for the 1D bar

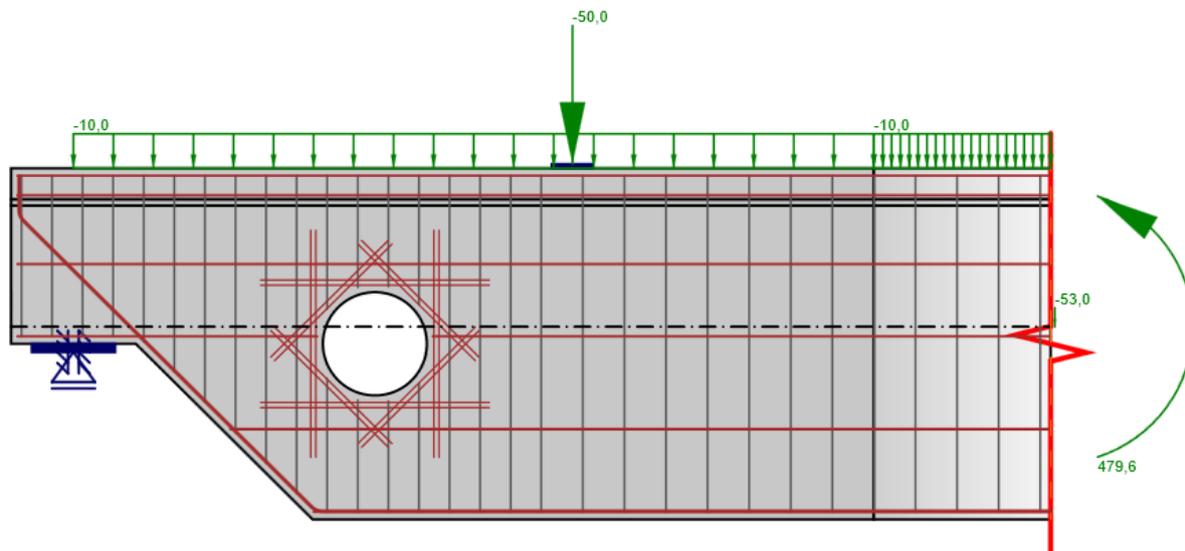


Fig. Example of load by point and line loads + internal forces of the cut

These loads must be defined in load conditions. The application distinguishes between load conditions of the following types

- Stable
- Random

Only the ultimate limit state (ULS) can be assessed separately for the load condition. Load condition combinations are defined by the guideline and the combination type. Types of combinations vary according to the selected standard. For EN, the following are available

- ULS (basic) combination
- SLS characteristic combination
- SLS quasi-permanent combination

For the combinations, the ultimate load and serviceability limits can be assessed. The availability of the calculation types analysis depends on the defined combinations and the selected assessment standard.

3 Reinforcement

Reinforced bars are modelled including the effect of tensile strengthening (more accurately reinforcement) of 1D bars (CROD element). Very important parts of the model are elements that provide strong links between the deformation parameters at the interconnected points of the network. Each link expresses one or more dependent degrees of freedom as a linear function of independent degrees of freedom. Three types of links are implemented, (i) rigid RBE2, (ii) the interpolation element RBE3, and (iii) the MPC link between a group of nodes. The RBE3 element can also be used for load distribution to the model. MPC is used to map the deformation from the concrete to the reinforcing network, see Fig. FEM network with reinforcement and concrete connection. All elements are implemented internally (non-user in the model), the application generates the model automatically.

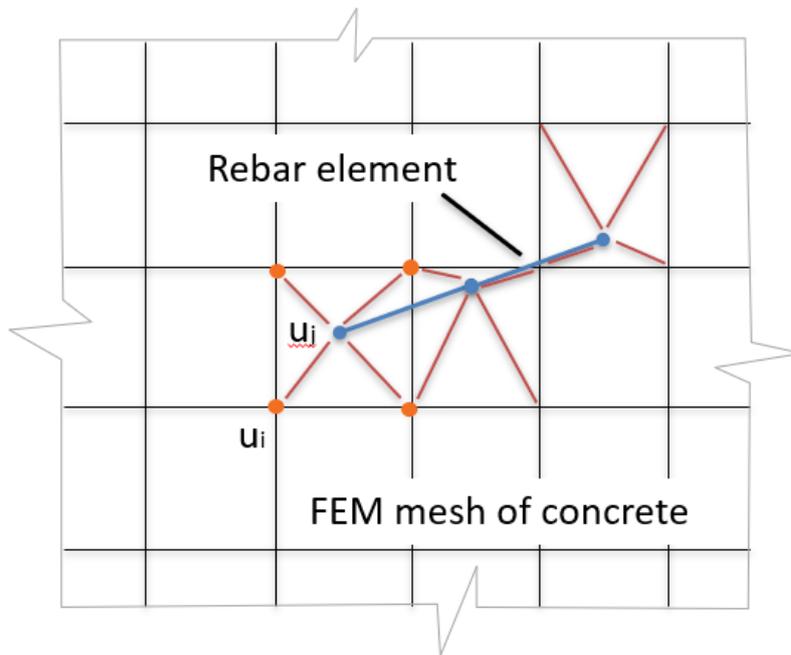


Fig. FEM network with reinforcement and concrete connection

A special feature of the new method is a specially developed model of cohesion and anchoring, see Fig. Cohesion and anchoring model. A special element of cohesion has been developed to ensure the interaction of reinforcing bars and concrete membrane (wall) elements. The stress on the cohesion element is governed by the bilinear dependence on the slip between the reinforcement insert and the concrete. Anchoring is modelled by additional rigidity (spring) at the end of the insert. The rigidity of the spring differs according to the type of finishing. The anchor force is calculated based on the rigidity of the spring with the limit value given by the ultimate strength of the concrete in coherence.

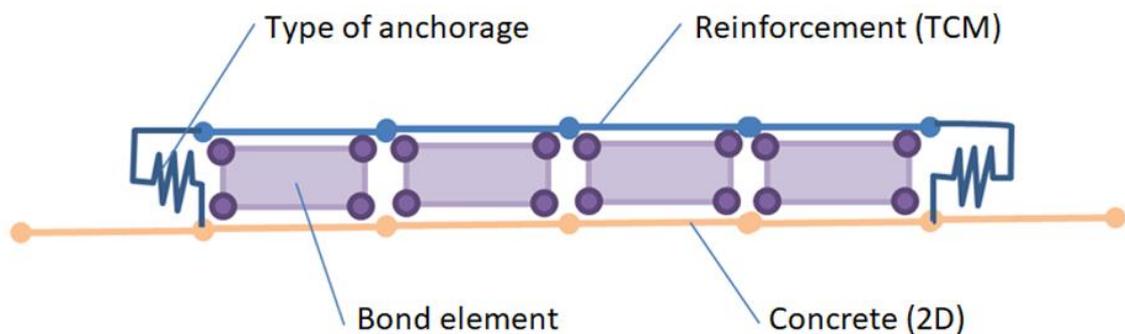


Fig. Cohesion and anchoring model

Concrete steel behaviour is basically described by an idealized bilinear work diagram, also known by design standards. In order to take into account the effect of the tensile reinforcement, a modification of the working diagram is introduced for the reinforcing inserts embedded in the concrete. The tension reinforcement model distinguishes cases of stabilized and non-stabilized crack distribution. In the case of fully developed stabilized cracks, the so-called Tension Chord Model (TCM) is used to express tensile strength. Considering that for the TCM, the tensile reinforcement depends on the reinforcement degree and on its assignment to each reinforcement insert or layer, the determination of the respective effective drawn (cooperating) concrete area is essential. For this reason, the

automatic spatial (3D) identification of the corresponding effective drawn concrete area for any reinforcement configuration was implemented.

Chapter text and images are drawn from [5].

3.1 Possibilities of reinforcement

3.1.1 Wall reinforcement

The reinforcement is inserted into the wall in its true position, respecting the thickness of the wall. For the calculation model, the reinforcement is transformed into the centreline of the wall, and its eccentricity in the perpendicular direction to the plane of the plate centreline is neglected. In case the item has multiple bars placed in a row, the inserts are transformed into the calculation model only as one bar with a modified diameter corresponding to the number of inserts in the layer.

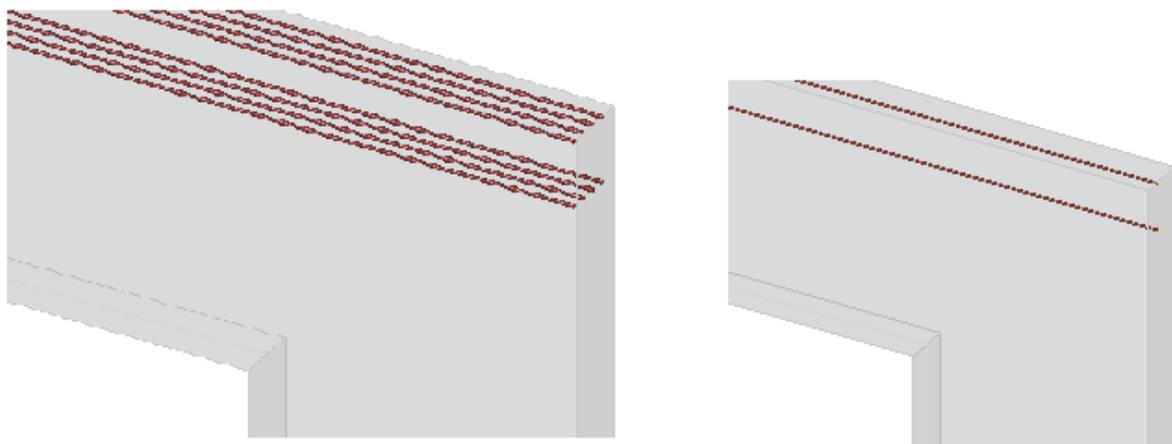
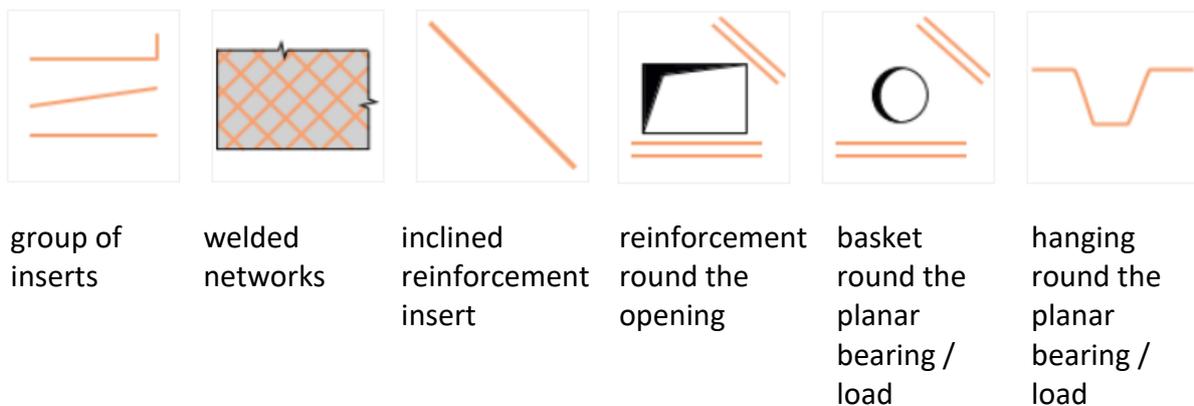


Fig. Real reinforcement calculation model of reinforcement in the wall

The location of the reinforcement depends on the covering given. The distance between the inserts in one row is determined automatically.

For reinforcement walls, the following reinforcement options are available:



3.1.1.1 Inserts group

The group of inserts means one or more reinforcing inserts at one wall surface. The groups of inserts can be defined only in the wall axis, at both surfaces simultaneously or, if necessary, several groups of inserts in the thickness of the wall. Reinforcement inserts can be input as straight bars using absolute or relative coordinates. Other input options are direct inserts defined to the wall edge or bent inserts defined at multiple edges of the wall.

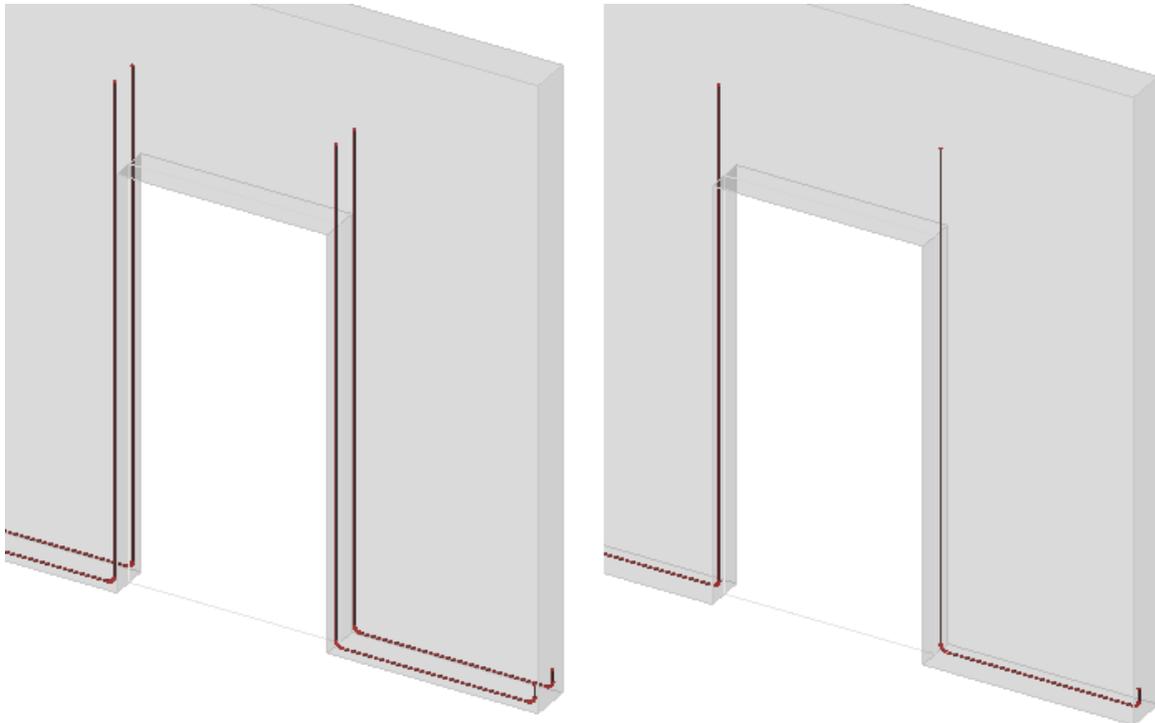


Fig. Real reinforcement set on multiple wall edges and a calculation model

3.1.1.2 Welded networks

The walls can be reinforced in the whole area by means of welded networks. There is a network automatically trimmed at the openings. The precise position of individual network inserts can be influenced by moving or rotating of the entire network. Similarly to the group of inserts, it is possible to define the net only in the centreline of the wall, on both surfaces, or even in multiple layers in the wall.

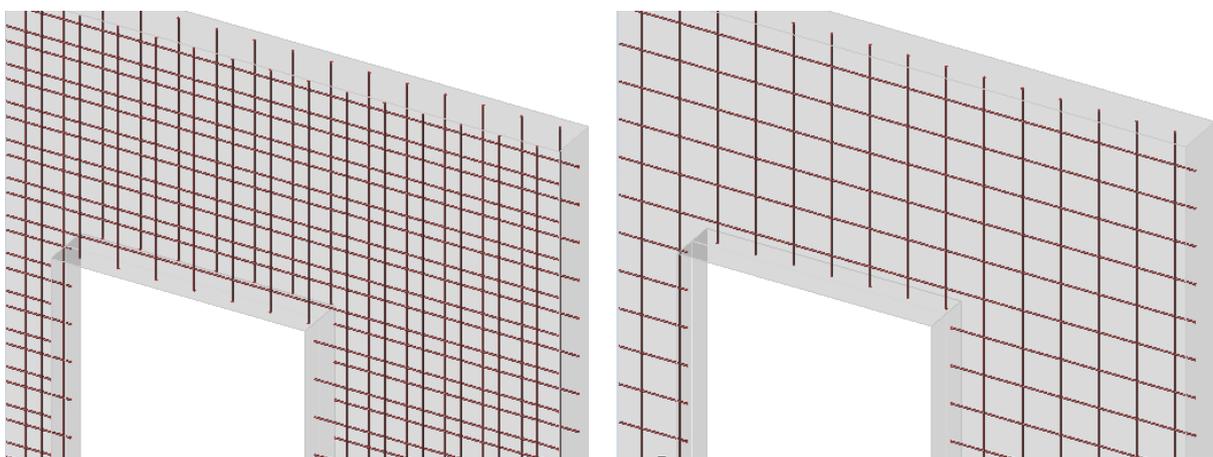


Fig. Reinforcement of network wall on both surfaces and its calculation model

3.1.1.3 Inclined reinforcement bars

Inclined reinforcement inserts allow quick insertion of the reinforcement on the concave corners of the structure (in case of the wall, on the corner of the window opening).

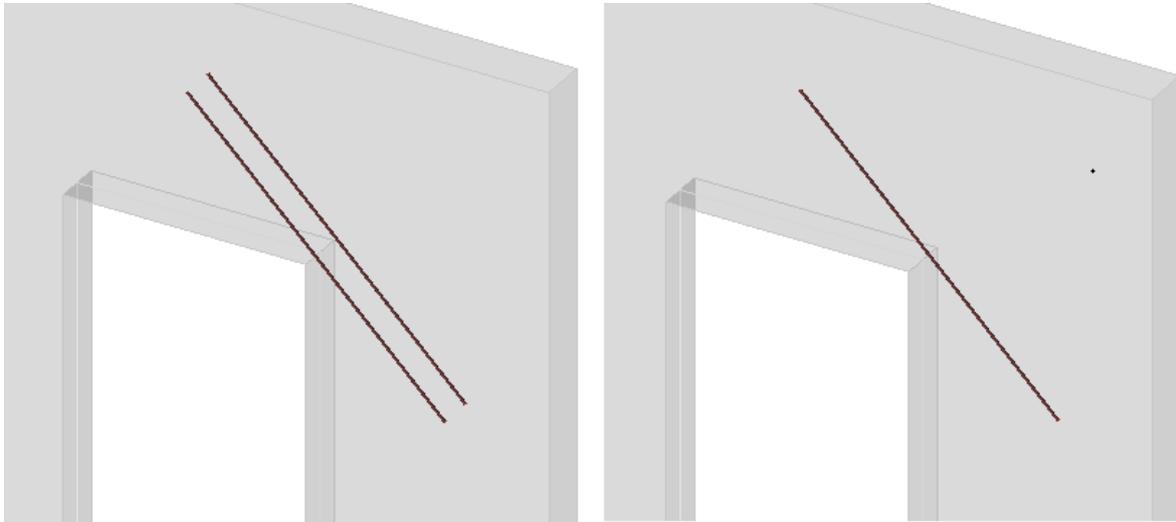


Fig. Inclined inserts at the door opening and their calculation model

3.1.1.4 Reinforcement round the opening

A special reinforcement basket is prepared to form the reinforcement around the opening. The reinforcement can be input around all the edges of the opening. It is also possible to define the diagonal reinforcement of the opening corners.

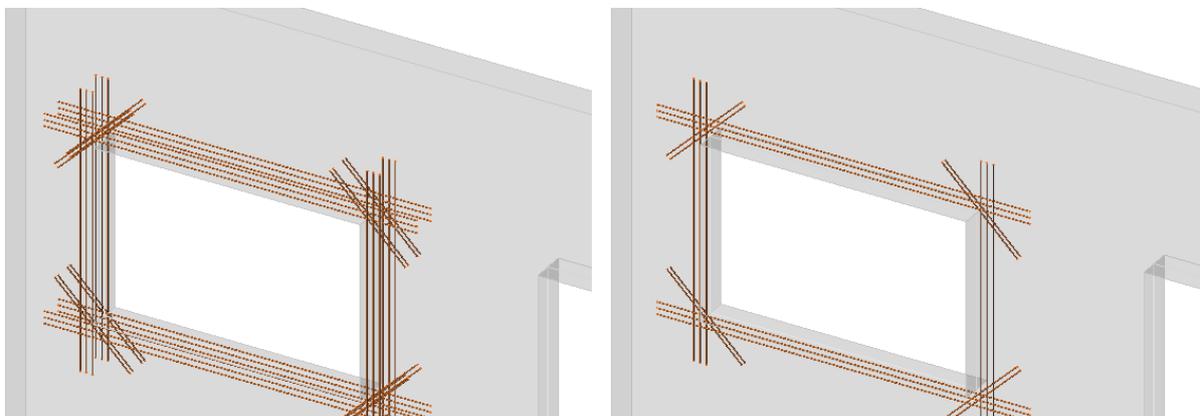


Fig. Reinforcement basket around the opening and its calculation model

3.1.1.5 Reinforcement round the planar bearing or load

To reinforce the planar bearing or load, a similar reinforcement can be used as for the reinforcement around the opening. The second option is to use hanging.

Reinforcement allows a transfer of bearing or load into the calculation model.

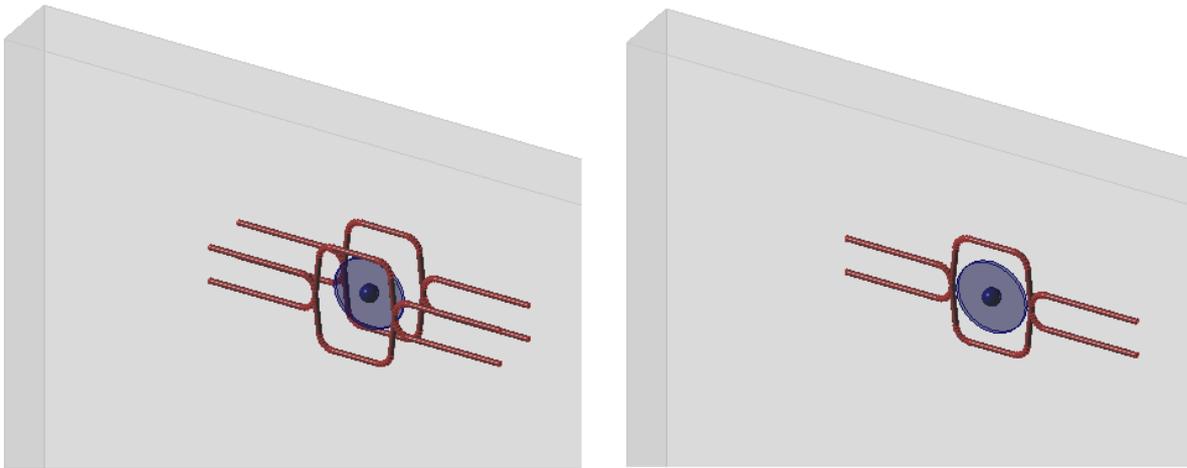
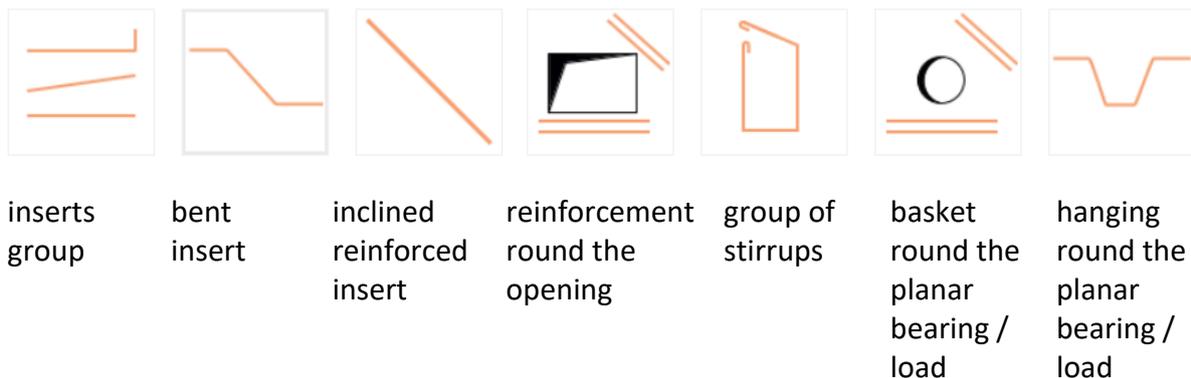


Fig. Reinforcement around point bearing and its calculation model

3.1.2 Reinforcement of beams

For the reinforcement of beams the same rules are applied as for the wall reinforcement. In the case of beams, the reinforcement is also transformed into the centreline of the replacement wall model. In the case of stirrups, the number of stirrup cuts is determined by the replacement diameter of the reinforcement bar in the model.



3.1.2.1 Group of stirrups

In the current version, stirrups are simply input by the number of cuts. The shape of closed stirrups is derived from the narrowest cross section and is always rectangular. When assembling a model, the stirrup branches acting perpendicular to the plane of the replacement wall model are neglected.

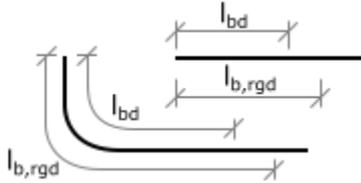
3.2 Finishing of reinforcement

Finishing of the reinforcement is modelled by additional rigidity at the end of the insert. The reinforcement is not extended in the model by the specified anchor length, the type of finishing is only determined to determine the additional rigidity at the ends of the insert. Types of reinforcement finishing are prepared according to [1] 8.4 and 8.5.

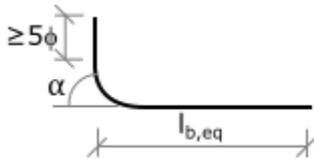
3.2.1 Finishing of longitudinal reinforcement for EC2

For the longitudinal reinforcement, the following types of finishing can be used according to EC2:

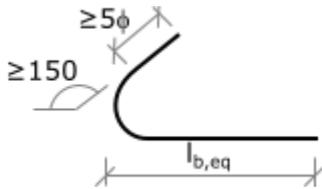
- No finishing (according to [1] Fig. 8.1 a). Additional rigidity at the end of the insert is zero.



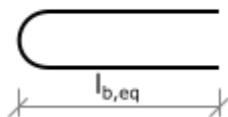
- Standard bend (according to [1] Fig 8.1 b). The additional rigidity at the end of the insert is expressed by a factor of 0.3. The radius of rounding is determined from the minimum radius of rounding in accordance with the profile used according to [1] 8.1N. The bend length is only used for the material report and is set to the minimum allowed value



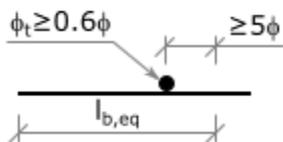
- Standard hook (according to [1] Fig 8.1 c). The additional rigidity at the end of the insert is expressed by a factor of 0.3.



- Standard loop (according to [1] Fig 8.1 d). The additional rigidity at the end of the insert is expressed by a factor of 0.3.



- Cross welded bar (according to [1] Fig 8.1 e). The additional rigidity at the end of the insert is expressed by a factor of 0.3.



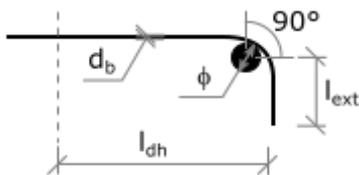
- Fully anchored insert, i.e. an insert where a complete cohesion with concrete throughout its length from the beginning or end of the insert is considered. Full anchoring can be accomplished, for example, by welding on a steel plate. Additional rigidity at the end of the insert is expressed by a factor of 1.0.

- Continuous insert, i.e. an insert that intersects the circumference of the model and where it is believed to continue on the cut part of the beam or wall model. As with a fully anchored insert, complete cohesion with the concrete is assumed throughout its length. Additional rigidity at the end of the insert is expressed by a factor of 1.0.

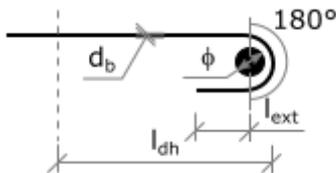
3.2.2 Finishing of longitudinal reinforcement for ACI

For the longitudinal reinforcement, the following types of finishing can be used according to ACI:

- 90 ° hook. The additional rigidity at the end of the insert is expressed by a factor of 0.3.



- 180 ° hook. The additional rigidity at the end of the insert is expressed by a factor of 0.3.



- Fully anchored insert with the same parameters as for the Eurocode.

- Continuous insert with the same parameters as for the Eurocode

3.2.3 Finishing of stirrups

The stirrup is modelled in the calculation model as a single bar with a replacement diameter based on the number of cuts specified. Finishing is always applied at the top part of the stirrup. Additional rigidity at the lower end of the stirrup is always expressed in the case of closed stirrups by the factor of 1.0 (it is actually a continuous, fully anchored bar). At the upper end of the stirrup, the value of the rigidity is expressed by the weight diameter according to the number of cuts and the selected finishing type.

3.2.3.1 Finishing of stirrups for EC2

The following types of finishing can be used for stirrups:

- Hook (according to [1] fig 8.5 a). The additional rigidity at the end of one branch of the stirrup is expressed by a factor of 1.0.



- Bend (according to [1] Fig 8.5 b). The additional rigidity at the end of one branch of the stirrup is expressed by a factor of 1.0.



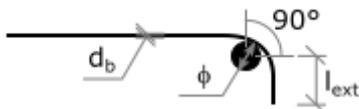
- Overlap. The additional rigidity at the end of one branch of the stirrup is expressed by a factor of 1.0.



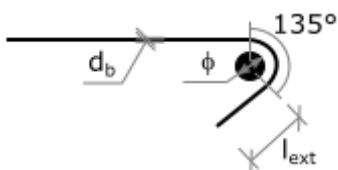
3.2.3.2 Finishing of stirrups for ACI

The following types of finishing can be used for stirrups:

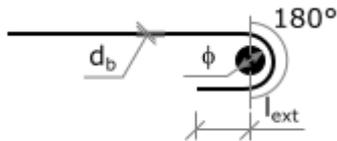
- 90 ° hook. The additional rigidity at the end of one branch of the stirrup is expressed by a factor of 1.0.



- 135 ° hook. The additional rigidity at the end of one branch of the stirrup is expressed by a factor of 1.0.



- 180 ° hook. The additional rigidity at the end of one branch of the stirrup is expressed by a factor of 1.0.



- Overlap. The additional rigidity at the end of one branch of the stirrup is expressed by a factor of 1.0.



3.3 Topological optimization

IDEA StatiCa Detail offers the possibility of designing the reinforcement layout using the topological optimization method from [2], which enables the identification of the optimal storage locations and reinforcement directions. By computing while the topological optimization the geometry of the replacement truss is automatically generated using only a certain percentage of the original volume of the structure. The resulting geometry can be used as a relatively accurate guiding tool for identifying areas that are loaded by pull or pressure only. This process does not differ from the truss analogy method, but using the topological optimization method, the truss is determined automatically.

3.3.1 Analysis

The principle of analysis is to maximize the overall rigidity of the search structure for a given load set. This is equivalent to minimizing the strain energy that is equal to the work done by the external load. The optimized structure is searched by an iterative algorithm, a solid concrete structure with homogeneous material density is taken as an initial estimate of geometry of the structure, and the density of each finite element varies in individual steps of iteration in a way that reduces total strain energy. In the calculation model, the rigidity of the element is proportional to its density. The 100% density element has its full original rigidity, while the 0% density element has zero rigidity. As a result, after reaching the optimum, only 100% or 0% elements can be interpreted as places where the material is present or not present.

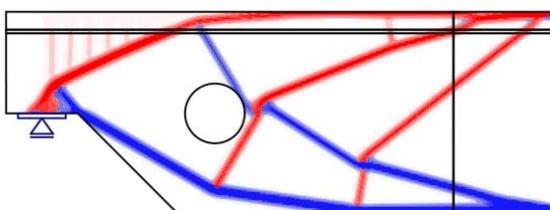


Fig. An effective volume of 20%

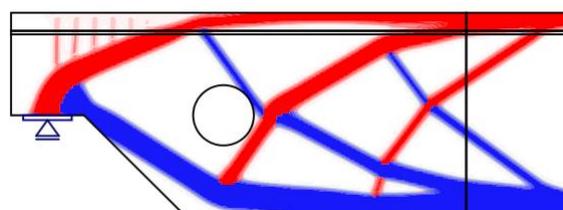


Fig. An effective volume of 40%

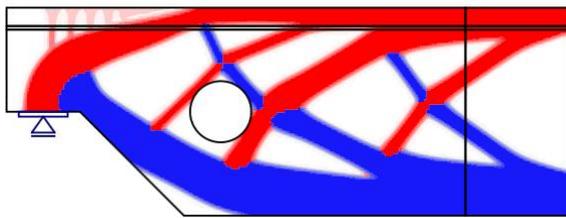


Fig. An effective volume of 60%

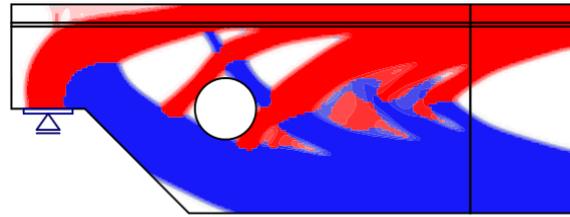


Fig. An effective volume of 80%

3.4 Linear analysis

Another possible reinforcement layout design is a linear analysis. After analysing the wall model, the areas of effective struts derived from the compression zones and the positions and directions of the rods derived from the main tensile stresses are shown on the construction that can serve as a guide for the positioning and reinforcement directions design.

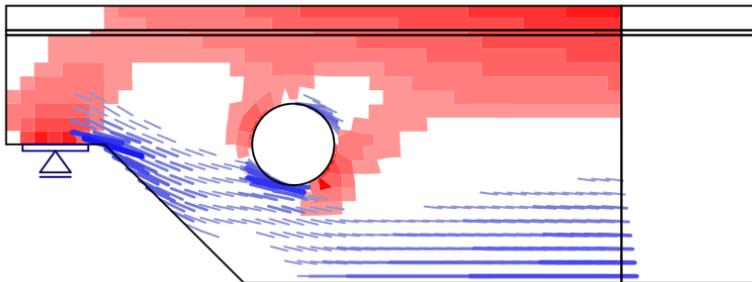


Fig. Design of struts and rods by linear analysis

3.5 Optimization of reinforcement of areas

For reinforcement profile design and number of inserts, IDEA StatiCa Detail application provides a tool for optimizing the area of the reinforcement. The iterative calculation optimizes the area of selected inserts so that the total weight of the reinforcement is as small as possible and at the same time that certain criteria such as reinforcement stress, concrete stress or maximum deformation of the structure are met. The resulting areas correspond to the required reinforcement areas and are presented to the user in the form of the required number of the current profile so as to make the reinforcement as efficient as possible. It also shows the required area and utilization (ratio of the required area to the current area).

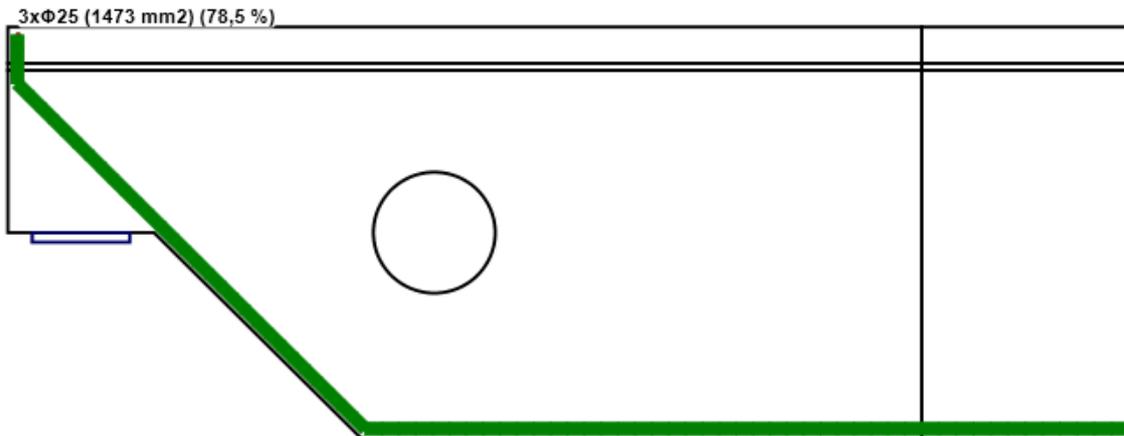


Fig. 1 Optimization of the area of the longitudinal reinforcement

The optimization of the stirrups is carried out by the so-called shear zones being generated from the selected reinforcements. The algorithm takes into account areas with a constant distance between the stirrups and the maximum length of the zone of 1.0 m. The calculated required area is then recalculated to a regular meter. For each zone, the required number of stirrups currently available (current number of cuts, current profile) and the conversion to the desired distance are specified so that the zone is most efficiently met. The necessary area of the stirrups is also presented on the regular meter of the zone and the utilization (ratio of the required reinforcement to the regular reinforcement → again to the regular meter).

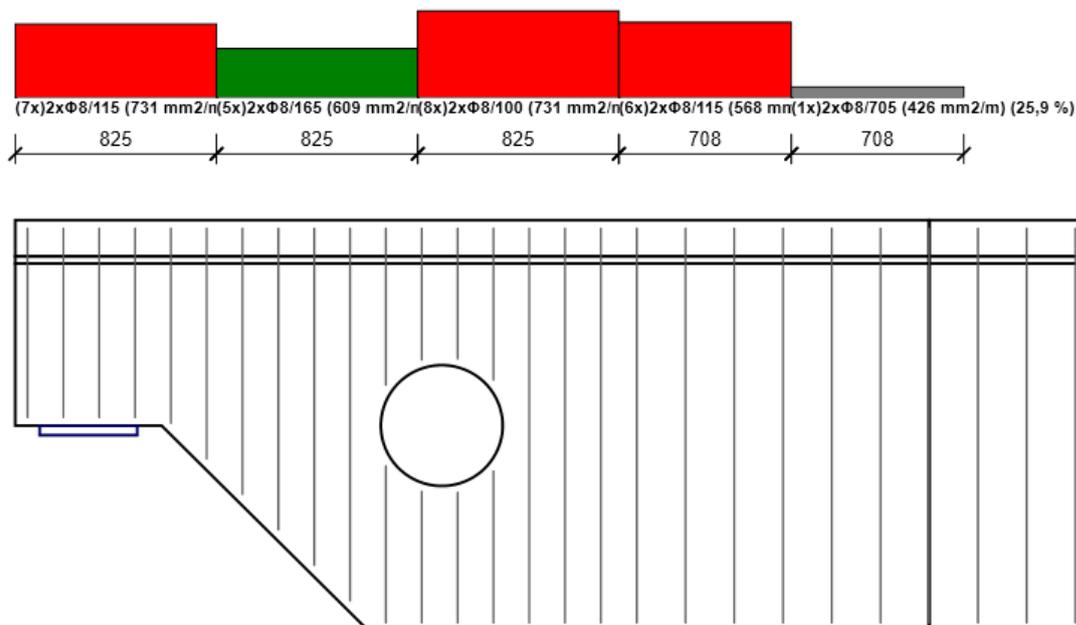


Fig. 2 Optimization of the stirrup area

4 Analysis

4.1 Physical model

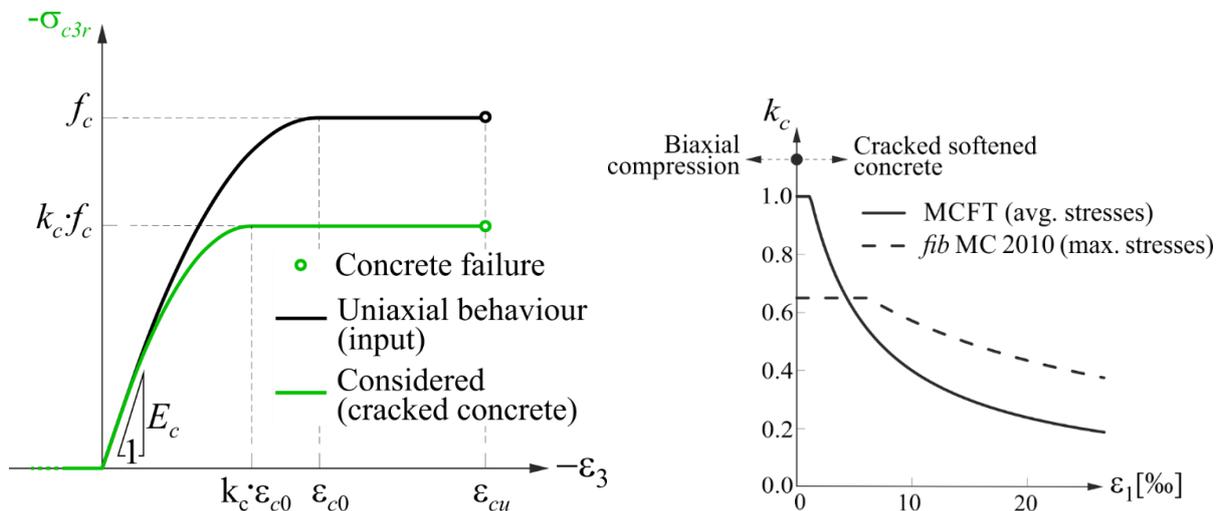
From a physical point of view, the method is based on a model of pressure fields with rods, assuming fictitious rotational cracks without stress in concrete in tensile (the tensile action of the concrete is modelled by tensile strengthening of a rod), zero incline, average

proportional strain and maximum tensile stresses in the crack, where there is also maximum stress in the concrete. The directions of the main stresses and the main strains coincide, which causes the separation of the behaviour in both main directions with the exception of the pressure softening effect (reduction of the compressive strength of the concrete due to the cross tensile strain) after the cracks have been created, the pressure softening is automatically calculated in dependence on the biaxial tension at that point.

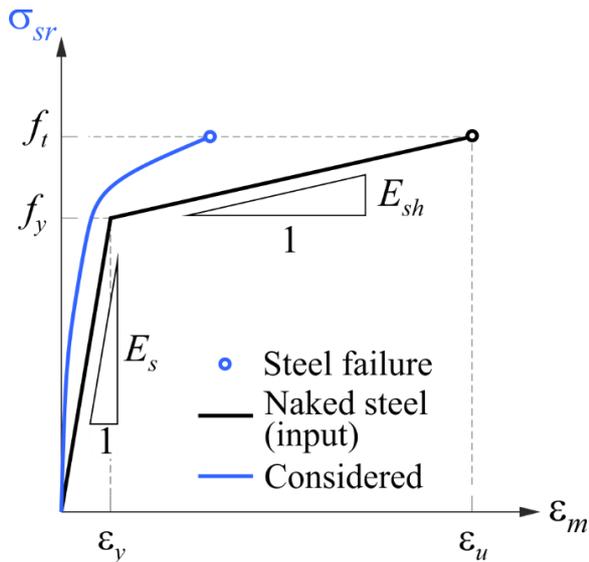
The constitutive relations for concrete are based on a standard parabolic-rectangular working diagram in uniaxial stress, which is known from [1] and derives only from the class of concrete, or from its compressive strength. Concrete steel behaviour is basically described in the default setup by an idealized bilinear work diagram, also known by design standards. In order to take into account the effect of the tensile reinforcement, a modification of the working diagram is introduced for the reinforcing inserts embedded in the concrete. The tensile reinforcement model distinguishes cases of stabilized and unstabilized crack distribution. In the case of fully developed stabilized cracks, the so-called Tension Chord Model (TCM) is used to express tensile reinforcement. Considering that in the TCM tensile reinforcement depends on the reinforcement stage and on its assignment to each reinforcement insert or layer, the determination of the respective effective drawn (cooperating) concrete surface is essential. For this reason, the automatic spatial (3D) identification of the corresponding effective drawn concrete area for any reinforcement configuration was implemented.

4.1.1 Physical model – ultimate limit state

The working diagram of the concrete is used as described in 5.1. A bilinear diagram can also be used. In the case of a large cross tensile ϵ_{s2} , the diaphragm is softened in the pressure part. The relation between the ϵ_{s2} value and the softening factor k_{c2} is described in the diagram. For simplification, the value of $1 / k_{c2}$ is set.

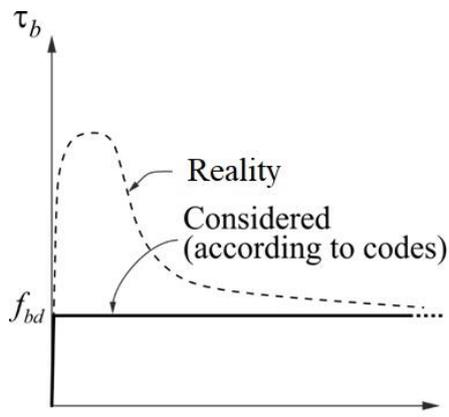


Concrete reinforcement is described in 5.1. Depending on the type of reinforcement, either the TCM or the so-called POM (Pull Out Model) is used. POM is normally used on stirrups if the basic percentage of the geometric degree of reinforcement exceeds 0.6 %. Otherwise, the TCM model is also used but with other parameters of the reinforcement degree.



The working diagram of so-called bearing plates is purely linear and not limited. It is an auxiliary model entity that is not evaluated.

For defining the physical properties of concrete and reinforcement modelling elements in coherence (see 5.2), the values are used which may vary in each final 1D reinforcement element depending on the reinforcement position in the structure. The values such as the material characteristics of concrete at the given place and the position of the reinforcement in dependence on the direction of concreting are used.



Similarly, physical properties are obtained for the reinforcement finishing elements.

4.1.2 Physical model – serviceability limit state

Unlike in 5.1.1, a bilinear diagram with infinite branches is used here. The analysis is performed on two models differing only in the rigidity of the diagram branch for the long-term load component. The module E value for the long-term load component is reduced by the creep coefficient value. This value is entered directly into the application and is not analysed.

The concrete reinforcement model used is the same as in 5.1.1, except that the first and last branch of the diagram is not limited.

For the material of the bearing plates the information is described in 5.1.1

4.2 Mechanical model

The basis of the model is a net of finite elements (predominantly of square shape) on the concrete part. This is a wall 2D model. As described in 2.1. The basic element size depends on the size of the entire detail and its parameters are set in the setup.

As described in 4, concrete reinforcement is modelled with 1D elements. The end nodes of these elements are not directly connected to the network of 2D elements of the concrete part, but by the so-called MPC links. For the calculation of the serviceability limit state, these links are directly linked between 1D and 2D elements. For ULS, auxiliary 2D elements are inserted between them to model cohesion between concrete and concrete reinforcement. At the same time, a special 1D element is inserted at the end of the bar, which models the finishing of the reinforcement.

Boundary conditions are modelled by entities that take a defined degree of freedom. In the case of point bearing, a null shift or rotation is defined at that node. Depending on the type of bearing, this removed degree of freedom is transferred to the concrete part, either by means of a bearing plate or directly with the estimation on the network nodes near the bearing node. In the so-called line bearing, it is transferred directly to the edge nodes of the network of concrete elements. This is not a rigid link, but there are elements that model the rigidity of bearing. For the linear bearing, the so-called nonlinear behaviour can be specified when the given bearing only transfers pressure.

The so-called bearing plates are modelled independently of the concrete part of the network. The network size is smaller than the size of the adjacent concrete part. Both networks are not connected directly, but by MPC links, where the nodes at the boundary of the bearing plate are connected to the nodes of the concrete part. The size of the area where it is made is dependent on the size, especially the thickness of the bearing plates.

The point load is either applied by means of bearing plates or the size of the area to which the load is applied - see 2.4.2

Line load is always applied to the nearest border nodes of the network, when applied to the edge, or to the nodes of the line on which it is specified.

4.3 Analysis

The non-linear analysis method is Newton-Rapshon. In the default setting, only one load increment is defined for the load state which is then presented. In practice, this means that for the load case itself the results of the last "transferred" load are presented and for the combination always for the "transferred" permanent component and then for the random. It is possible to customize the numbers of these displayed results both for the permanent component and for the random.

In the case of setting of the analysis termination when reaching the limit strain on any element of the structure (default) and not transferring the total load specified, at the same time this is signalled by the message.

5 Checks

5.1 Checks according to EC

The checks in IDEA StatiCa Detail are divided according to the ultimate limit state (ULS) and serviceability limit state (SLS).

The ultimate limit state includes the checks of:

- Concrete strength - based on the assumptions of the solution according to ULS 5.1, the stress on the finite element network is calculated, which is then compared with the limit stress defined in the working diagram of the concrete.
- Reinforcement strength - based on the assumptions of the solution according to ULS 5.1, the stress is calculated over the length of the reinforcement, this stress is then compared with the limit stress of the concrete reinforcement defined within the working diagram of the reinforcement.
- Anchorage (cohesion limiting stress) - based on the assumptions of the solution according to ULS 5.1, the reinforcement [4] and the finishing of the reinforcement, the stress in cohesion in the anchor area is calculated and this is then compared with the limit stress calculated on the basis of the profile, the surface of the reinforcement type and the shape of the anchor length.

Limit states of serviceability include the checks of:

- Stress limitation according to the 7.2 standard [1] - based on assumption of solutions 5.1 The physical model, the stress is calculated on the finite element network and compared with the limit stress of the concrete described in 7.2 (1) (2) [1]. In the stress limit check, the stress along the length of the reinforced inserts is also calculated and compared with the limit stress of the reinforcement 7.2 (3) [1].
- Crack check - the crack check analysis is divided into the analysis of stabilized cracks (evenly distributed - for example at the lower edge of the beam and non-stabilized (isolated), which occur at the places of high stress concentration (e.g. at sharp edges, dapped ends, shear cracks at walls). All stress and strain analysis is performed on the basis of assumptions 5.1.2 The crack analysis itself is performed using the methods described in [3]
- Strain (deflection) check - within deflection analysis - short-term deflections of the total load, deflection of long-term creep load, incremental deflection of short-term load, and total deflection are calculated. The analysis is performed against the limit values input by the user.

5.2 Checks according to ACI

Analysis of ultimate limit state according to ACI are identical to the ultimate limit state according to 6.1 Checks according to EC. Finishing of the reinforcement is taken for the analysis according to 4.2.2 and 4.2.3.2.

The analysis of the serviceability limit states is reduced only to the strain check analysis and is identical to the analysis according to 6.1

6 Literature

- [1] EN 1992-1-1 *Eurocode 2, Design of Concrete Structures – Part 1: General rules and rules for buildings*, European Committee for Standardization, December 2004-2016
- [2] KONEČNÝ, M., KABELÁČ, J., NAVRÁTIL, J. Use of topology optimization in concrete reinforcement design, In: *Sborník ke konferenci 24. BETONÁŘSKÉ DNY 2017*, Czech Republic, 2017
- [3] MATA-FALCÓN, J., TRAN, D., T., KAUFMANN, W., NAVRÁTIL, J. Computer-aided stress field analysis of discontinuity concrete regions, In: *Proceedings of EURO-C 2018 Computational Modelling of Concrete and Concrete Structures*, Austria, 2018, in print
- [4] NAVRÁTIL, J., ŠEVČÍK, P., MICHALČÍK, L., FOLTYN, P., KABELÁČ, J. Řešení stěn a detailů betonových konstrukcí, In: *Sborník ke konferenci 24. BETONÁŘSKÉ DNY 2017*, Czech Republic, 2017
- [5] NAVRÁTIL, J., ŠEVČÍK, P., KABELÁČ, J. Řešení stěn a detailů betonových mostů, In: *Sborník ke konferenci 23. MEZINÁRODNÍ SYMPOZIUM MOSTY 2018*, Czech Republic, 2018