Eidgenössische Technische Hochschule Zürich
Swiss Federal Institute of Technology Zurich

Chair of Concrete Structures and Bridge Design
Institute of Structural Engineering
D-BAUG, Master's degree in Civil Engineering
Autumn Semester 2026

Advanced Structural Concrete:

Workshop on the Compatibility-based Stress Field Method

(101-0127-00L)

Topic: Dimensioning with the software *Idea StatiCa Detail*

In this workshop, the basics for the use of the software *Idea StatiCa Detail* are introduced. *Idea StatiCa Detail* implements the Compatibility-based Stress Field Method (CSFM) presented in Chapter 2.6 of the course. The main focus of the software is on reinforced concrete structures with a source of static or geometric discontinuity (Discontinuity Regions), but it can also be used consistently in conventional regions where the Bernoulli hypothesis remains valid. The Compatibility-based Stress Field Method is implemented in *Idea StatiCa Detail* based on the finite element method, but it works according to the main assumptions of the stress field method of the theory of plasticity.

The workshop is composed of the following three tasks:

A) Installation of the *Idea StatiCa Detail* software (**PRE-TASK**)
B) Basic learning of the workflow and functioning of the *Idea StatiCa Detail* software by means of a step-by-step example (**PRE-TASK**)
C) Dimensioning of Discontinuity Regions by means of the Compatibility-based Stress Field Method

Students should complete the first two tasks before the workshop based on the guidelines provided in this document (this should take approx. 30-60'). The issues and questions that arose in these steps will be compiled in order to be discussed in plenum at the beginning of the workshop.

Detailed information for each task is provided below.

A) Installation of the *Idea StatiCa Detail* software (PRE-TASK)

To install the program, complete the following steps:

- Download and install the software: https://www.ideastatica.com/product-downloads
- Request free educational license (valid for 12 months) from: www.ideastatica.com/educational-license/
- In case of problems, additional information regarding the licensing can be found at: https://www.ideastatica.com/support-center
- If the problem persists, contact *Idea StatiCa*

B) Step-by-step example in the *Idea StatiCa Detail* software (PRE-TASK)

Several step-by-step examples are documented in the following link:
https://www.ideastatica.com/support-center/support-center-tutorials

Complete one of these online examples in order to get used to the interface of the program and detect questions/issues that will be discussed at the beginning of the workshop. An additional example of a dapped-end beam with an opening is documented below. This is a more advanced example that shows non-standard features of the program and proposed tasks. This example can be used as an alternative to the online examples, but it is more time-consuming.

Design of damped-end beam with/without opening

1. Introduction to the User Interface and First Steps

When opening the *Idea StatiCa* interface select the *Detail* program. When the *Detail* program is started, the user interface shown in Fig. 1 appears first. In the blue-framed area, you can create a new project or open an existing project. Under **Preferences**, you can manage the units, number of significant digits, as well as other preferences (not related to the mechanical model) and the license.

Check the settings for units and adjust them to your preferences

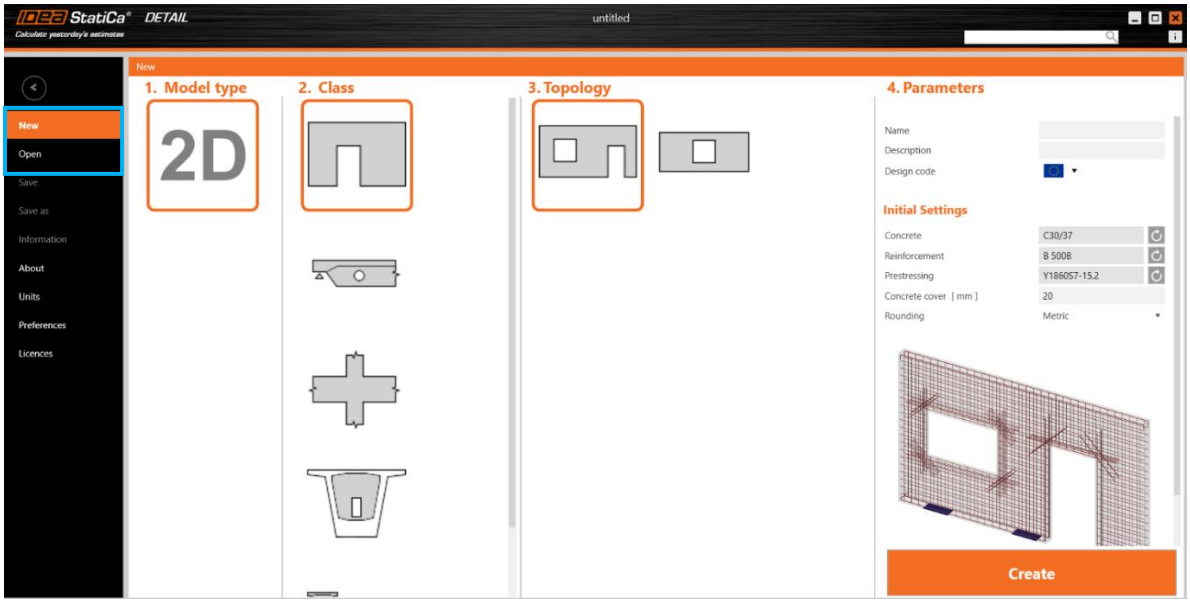


Fig. 1*:* User interface after starting

2. Modelling

2.1 Geometry

The T-beam shown in Fig. 2 is discussed in the following. It has a dapped-end on the left side. Therefore, the structural member is a discontinuity region. In addition to the variable loads shown in Fig. 2, the self-weight shall also be taken into account as a linear load on the upper edge of the girder (note: the self-weight is not automatically computed in the current version of the program).

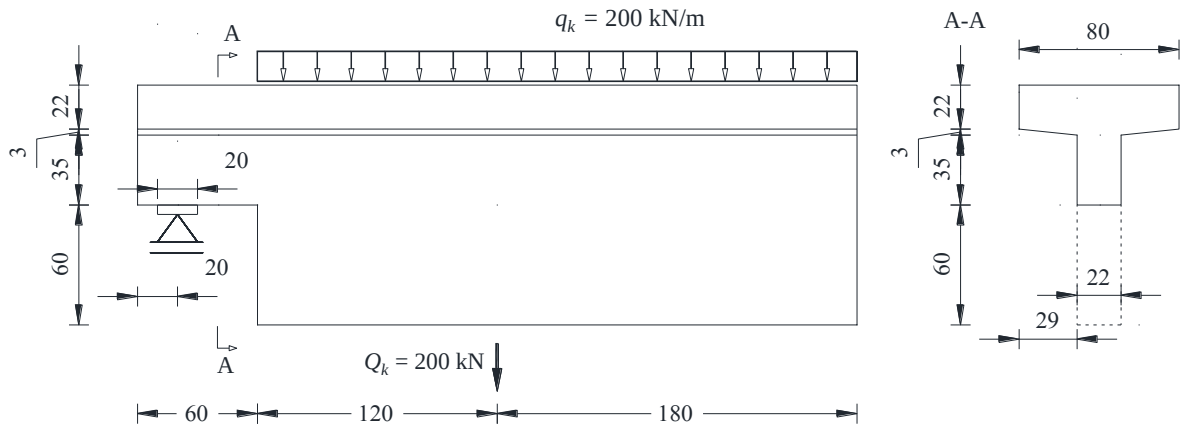


Fig. 2: Dapped-end T-beam with variable loads. Dimensions in cm

To start building the model, create a new project, and then select the **Beam cut-offs** option shown in Fig. 3a) (NB: the different templates lead to different input properties, but the resultant mechanical model is in all cases the same; if one has a different shape, one can use the general shape). Next, you can select the topology of the structure. Choose the **dapped end using different cross-sections (**see Fig. 3b)). A name and description for the project can also be given.

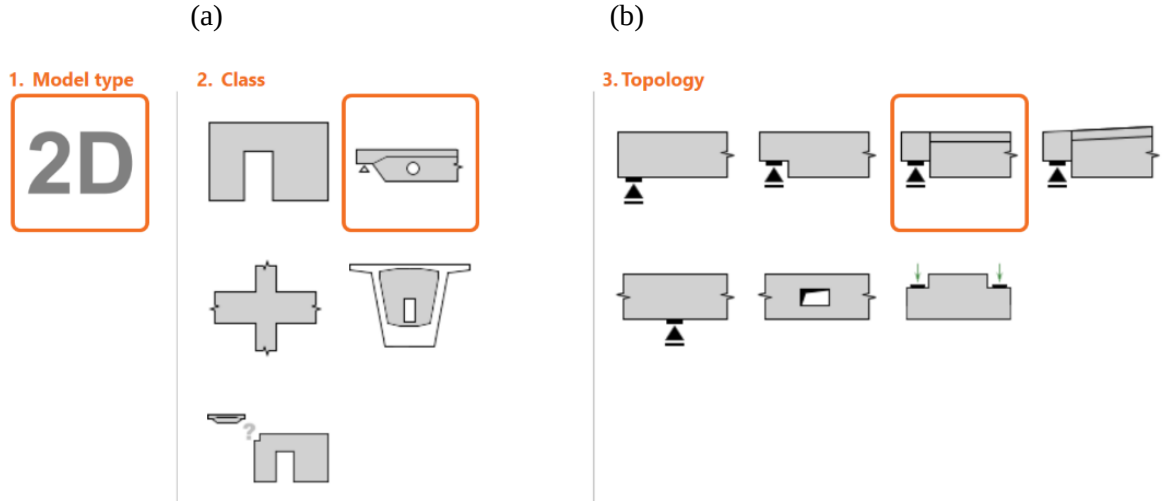


Fig. 3: (a) Discontinuity region class; (b) Topology of discontinuity region

When in the **DESIGN** tab, a dropdown menu is located on the right, providing an overview of members, supports, reinforcement, etc. In this navigator list, select the members "M1" and "M2" and modify their properties according to Table 1 and Fig. 2. You can modify the cross-section geometry and material using the **Edit** or **Add** button (see blue circle in Fig. 4 below).

Table 1: Geometrical description - Parameters to be defined

M1 beam	L – Length	[m]	0.6
	h	[mm]	600
	b	[mm]	800
	b_w	[mm]	220
	Concrete *	[-]	C35/45
M2 beam	L – Length	[m]	3.0
	h	[mm]	1'200
	b	[mm]	800
	b_w	[mm]	220
	h_f	[mm]	220
	Concrete *	[-]	C35/45

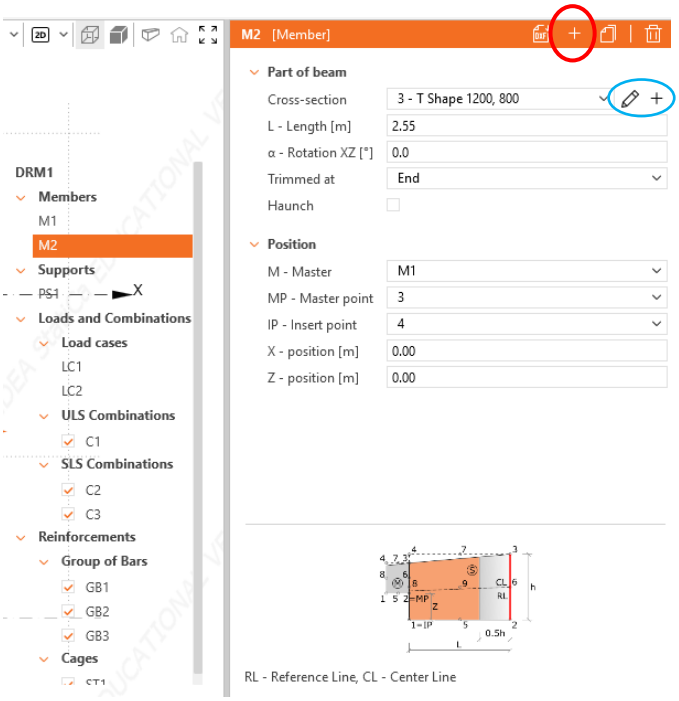
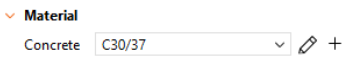


Fig. 4: Navigator list and editing options

* Tip:



Use the **Add** (+) button (see blue circle in Fig. 4) on the right to load a new cross-section from the library, which then appears in the drop-down list. Existing cross-sections can be modified manually using the pencil icon.

When an item from the navigator list is selected, using **Add** (+ button with red circle in Fig. 4) you can introduce further subregions or details ("material, openings, etc.). Adjust the point support PS1 according to Fig. 2 if needed.

Now it will be shown how to introduce the hanging punctual load by using the hanging load feature. Add a subregion and select the **Hanging load**. In the position tab, select the Masterpoint M2, the Bottom edge, and an x-position of 1.8 m from the end of the section (see Fig. 5). Select the type **L-shaped branches** (with branches perpendicular to related edge) and set the dimensions to L_1 = 0.20 m, L_2 = 1.0 m, L_3 = 0.40 m, and Ø = 20 mm.

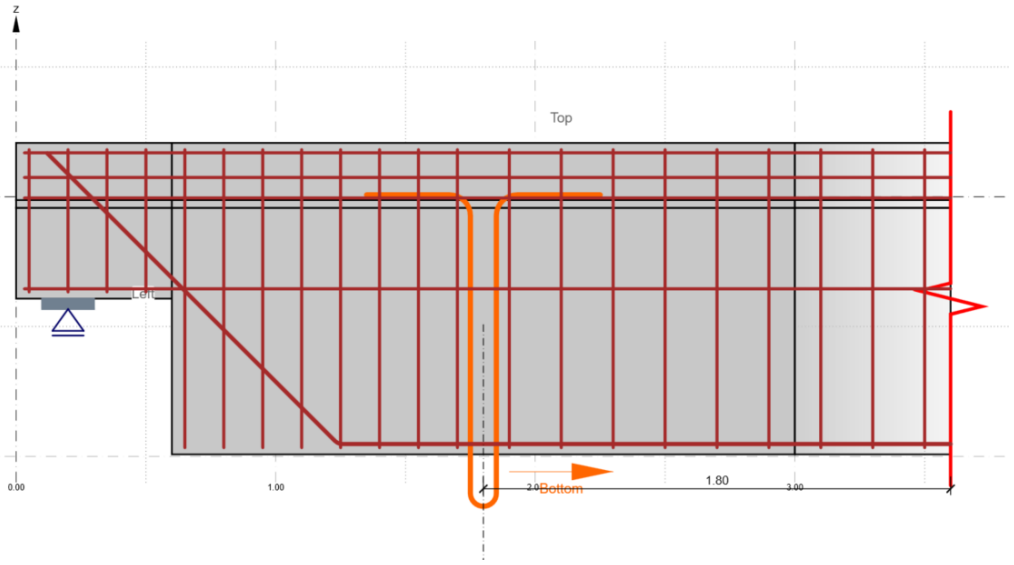


Fig. 5: Introduction of a hanging load

2.2 Loads

In the Navigator menu, switch to the next section **Loads and Combinations** to specify the loads (see Fig. 4)

NB: You can navigate freely in the navigator list, not following a sequential order. Additionally, you can navigate to the **MATERIALS** tab to specify further details, such as the material constitutive laws in more detail (this is a fundamental section). Furthermore, you can find summaries of the project, including the topology as a 3D model, all used cross-sections, general settings for modelling the reinforcement, as well as detailed compilations for reports (all these are more secondary tabs).

Dead load: Under the entry "Load Cases", the dead loads should be introduced as line loads in the dapped-end area as a load combination LC1 of permanent load type (line loads LL1 and LL2). Use the cross-sectional area data from Table 1 and a spatial self-load of the reinforced concrete of 25 kN/m^3.

Table 1: Cross-sectional area of the beam sections

A_{cs1}	0.2617 m^2
A_{cs2}	0.3937 m^2

Alternatively, the load could be applied in the middle depth of the beam.

Variable load:

Check if under the tab "Load cases" a load case LC2 for variable loads has already been created; otherwise, create one by clicking **New**. Choose the load case type "Variable".

A line load of -200 kN/m acts on subsection M2.

A single load of -150 kN is to be assigned to the load hanging device (HG1).

Internal forces:

In the tab "Internal Forces" adjust the support reactions for the load case LC1 (permanent loads) by setting the shear force at the right of the bearing plate (x = 0.2 m) to 25.0 kN (see Fig. 6). If you have defined the self-weight LL1 (for the dapped end) up to the left border, you need to adjust the left side of the bearing plate so that V = -1.3 kN and M = -0.1 kNm holds for the internal forces (equilibrium!).

Repeat the operation for the combination LC2. The support reaction in order to guarantee equilibrium should be set to 460.0 kN.

NB: Internal forces should be defined in this case, given that only part of the structure is modelled (strictly speaking, the introduced structural element is not in equilibrium, and inputting suitable internal forces that guarantee equilibrium is required). When modelling an entire structural element (the standard case), the definition of the internal forces is not required (only the definition of load impulses).

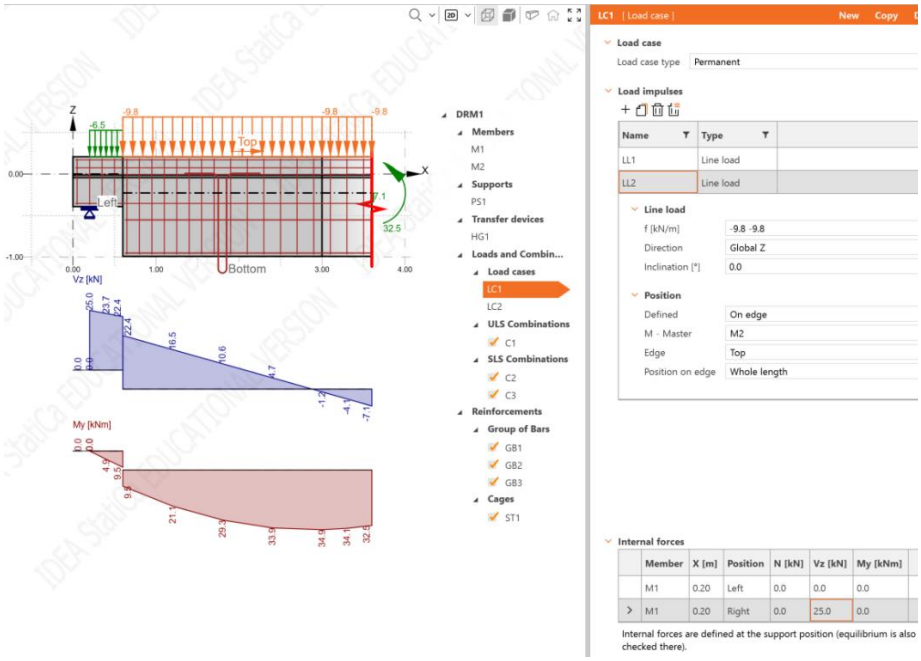


Fig. 6*:* Internal forces for permanent loads (LC1).

Load combination

The software should have automatically generated three load combinations. If this is not the case, the load combinations should be introduced using (+) or:



In the Ultimate Limit State (**ULS**), the permanent dead loads are combined with a factor of 1.35, and the variable loads are combined with a factor of 1.5. For the serviceability limit state (**SLS**), there are two checks: The characteristic loads for one (weight 1.0) and quasi-permanent loads (weight 1.0 for permanent loads and 0.3 for variable ones) for the other are chosen.

2.3 Reinforcement

If you now navigate to the **TOOLS** tab, you will see some design tools. **Topology Optimization** can be used to generate a potential flow of forces for the given geometry and load (see Fig. 7). This can be displayed for individual load cases or load combinations. The effective volume of the compression and tension areas can be defined.

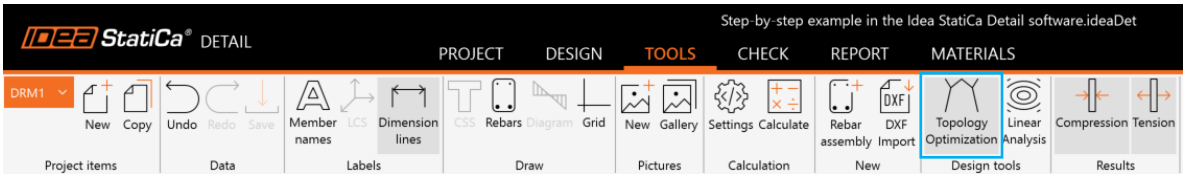


Fig. 7: Auxiliary design tools for the location of reinforcement

These design tools serve as qualitative help for the determination of the areas under tension, assuming a very simplified linear elastic behaviour. Therefore, designers are expected to interpret this information and provide a different location, adapted to the structural needs. Moreover, these tools do not provide any information on the required reinforcement content.

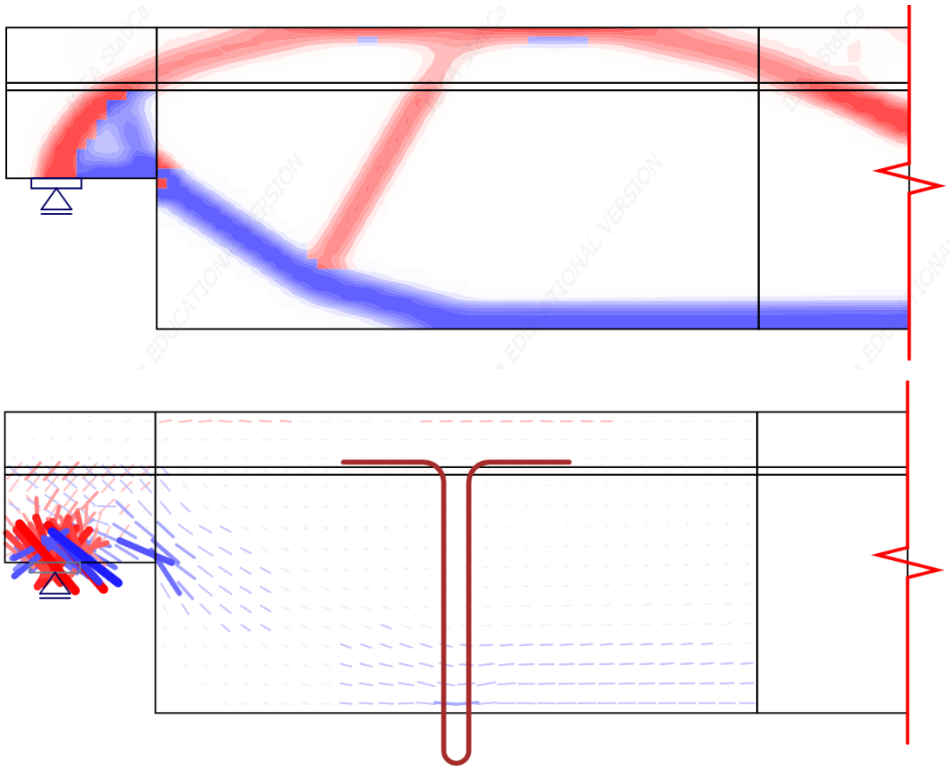


Fig. 8: Output of the topology optimization (top) and the linear analysis (bottom) with the corresponding visualization of the tension and compression fields.

In addition to Topology Optimization, **Linear Analysis,** a linear-elastic analysis of the structure, is also available as an auxiliary tool. Fig. 8 shows the results of both tools.

Think about where the reinforcement should be located. Go back to geometry, set the girder length from M2 to 4 m, and insert a 0.6 m x 0.6 m opening between 1.5 and 2.1 m. Repeat the analysis with the reinforcement location design tools and think about where the reinforcement could be located.

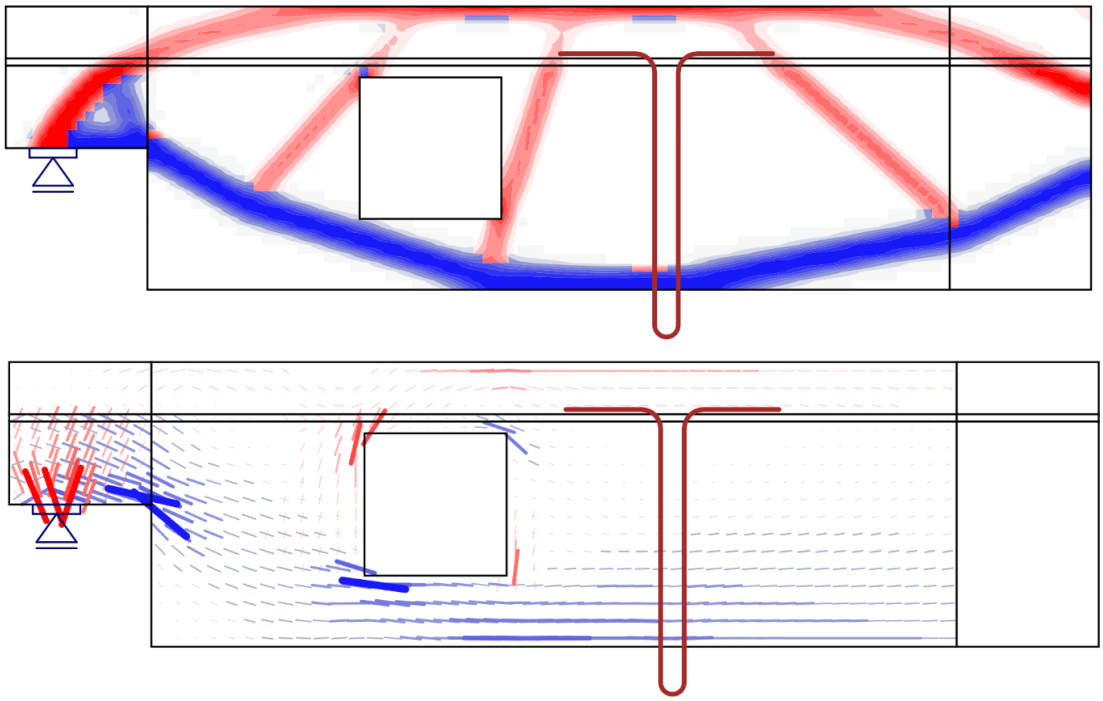


Fig. 9: Output of topology optimization (top) and linear analysis (bottom) for the case with an opening.

Undo the changes to the beam length and the opening and reinforce the beam. Make a short manual calculation to estimate the required bending reinforcement. Introduce the reinforcement based on the available reinforcement types displayed in Fig. 10.

Hint: Deactivate the load combinations for the serviceability limit state (Menu **Loads**). If necessary, increase the element size in the menu **Check** > **Settings** in order to faster determine the optimum reinforcement for the ultimate limit state iteratively. Once you have found a suitable reinforcement, you can reset the settings and start the structural analysis.

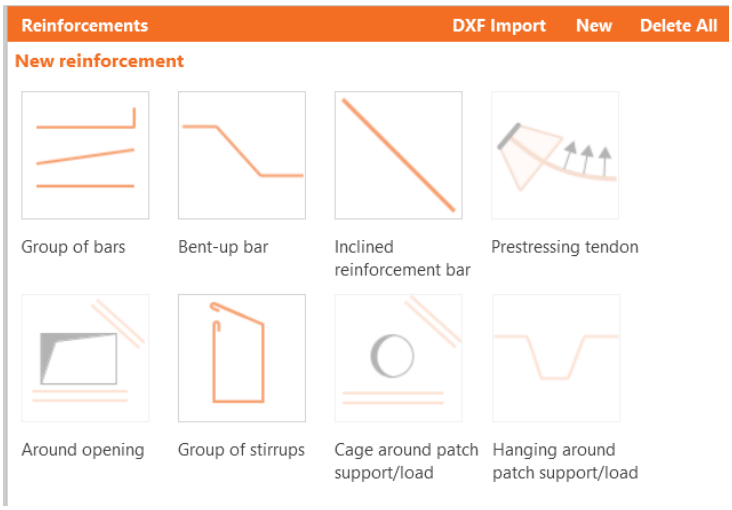


Fig. 10: Available reinforcement types for beams.

In order to complete the design of the structural member, complete the following tasks:

- Carry out the necessary verifications for the ultimate limit state.

- Carry out the necessary verifications for the serviceability limit state. Compare the crack widths with those of a short manual calculation. Use the tension chord model with the reinforcement content calculated by the program. Comment on the reinforcement content. Does the calculation make sense?

Excursus: Effective flange depth

For T-beams, the software accounts for the shear-lag phenomena with a conservative value of the reduced effective flange width. This is expressed by the parameter **Multiplier of flange depth (MFD)**. With the default value of 1.0, for example, a beam with the actual flange width of 960 mm will have an effective design width of 350 mm (see Fig. 11 b)). The 'Multiplier of flange depth' parameter can be changes by the user in the Check tab, under the Setup menu.

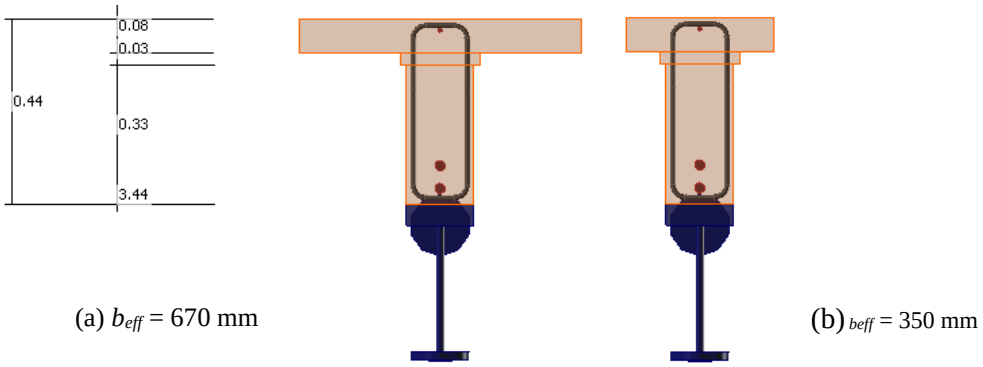


Fig. 11: Effective cross-section for a Multiplier of flange depth of (a) 3.0 and (b) of 1.0 for a 960 mm actual flange width.

The load deformation curves of the beams from Fig. 11 (with different MFD parameters) are shown in Fig. 12 for both parameters.

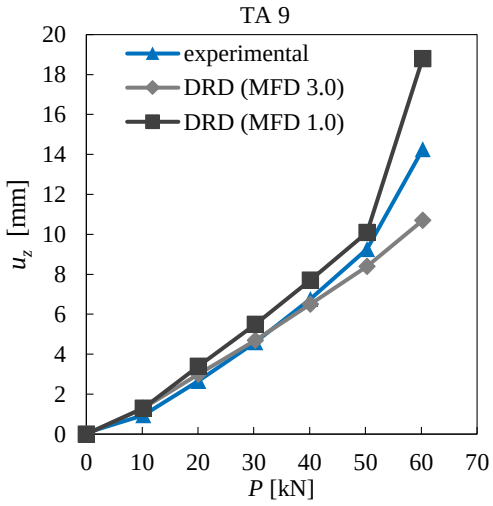


Fig. 12: Load deformation curves of the bending test TA 9 by Leonhardt & Walther (1963).

Literature

[1] Kommission SIA 262 (2003), "SIA 262 Betonbau." Schweizerischer Ingenieur- und Architektenverein, 2013.

[2] European Committee for Standardization (2004), "Eurocode 2: Design of concrete structures - Part 1-1: General rules and rules for buildings."

[3] Leonhardt, F., & Walther, R. (1962). *Schubversuche an einfeldrigen Stahlbetonbalken mit und ohne Schubbewehrung zur Ermittlung der Schubtragfähigkeit und der oberen Schubspannungsgrenze*. Ernst.

[4] Mata-Falcón, J., Tran, D.T., Kaufmann, W., Navrátil, J., *Computer-aided stress field analysis of discontinuity concrete regions,* Proceedings of the Computational Modelling of Concrete and Concrete Structures Conference (EURO-C 2018), Bad Hofgastein, Austria, 26 February – 1 March 2018, 641-650 pp.

C) Dimensioning of Discontinuity Regions by means of the Compatibility-based Stress Field Method

In the example, for the given design loads, do the detailing of (i) reinforcement, and (ii) supports and applied loads by using the Compatibility-based Stress Field Method. The given load is the most restrictive design load (no load combinations should be considered). For at least one region, check the deformation for a serviceability load of 50% of the given ultimate design load.

Materials in all cases: concrete C30/37 and reinforcement B500B

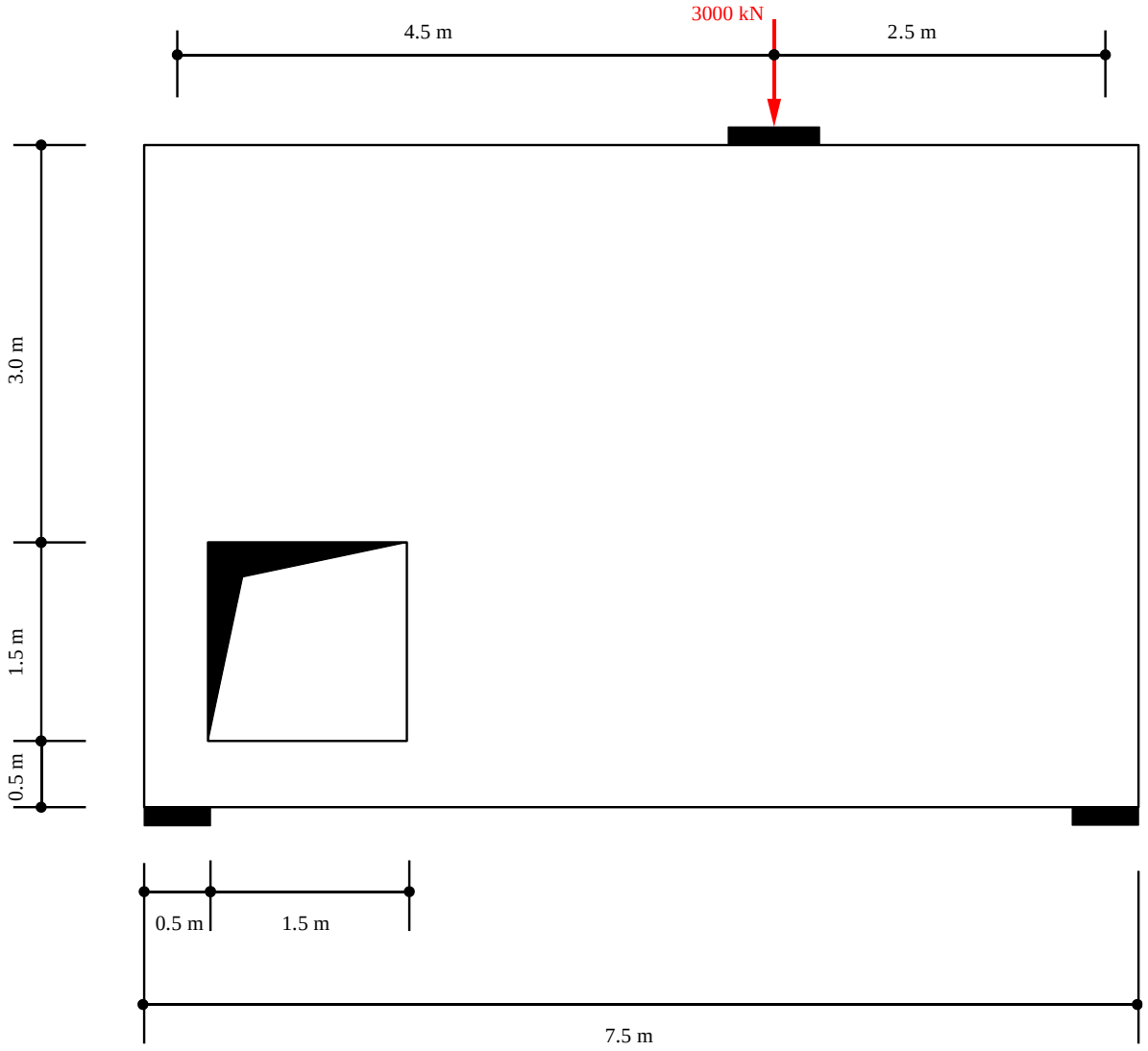


Fig. 13: Beam 2 with a wall thickness of 40 cm. Dimensions in cm.

In a first step, enter only the reinforcement necessary according to the strut-and-tie model. You can use the strut-and-tie model in Fig. 14. Analyse the results of the ULS check.

In a next step, include the minimum reinforcement (recommendation: 0.2 %). What changes do you observe?

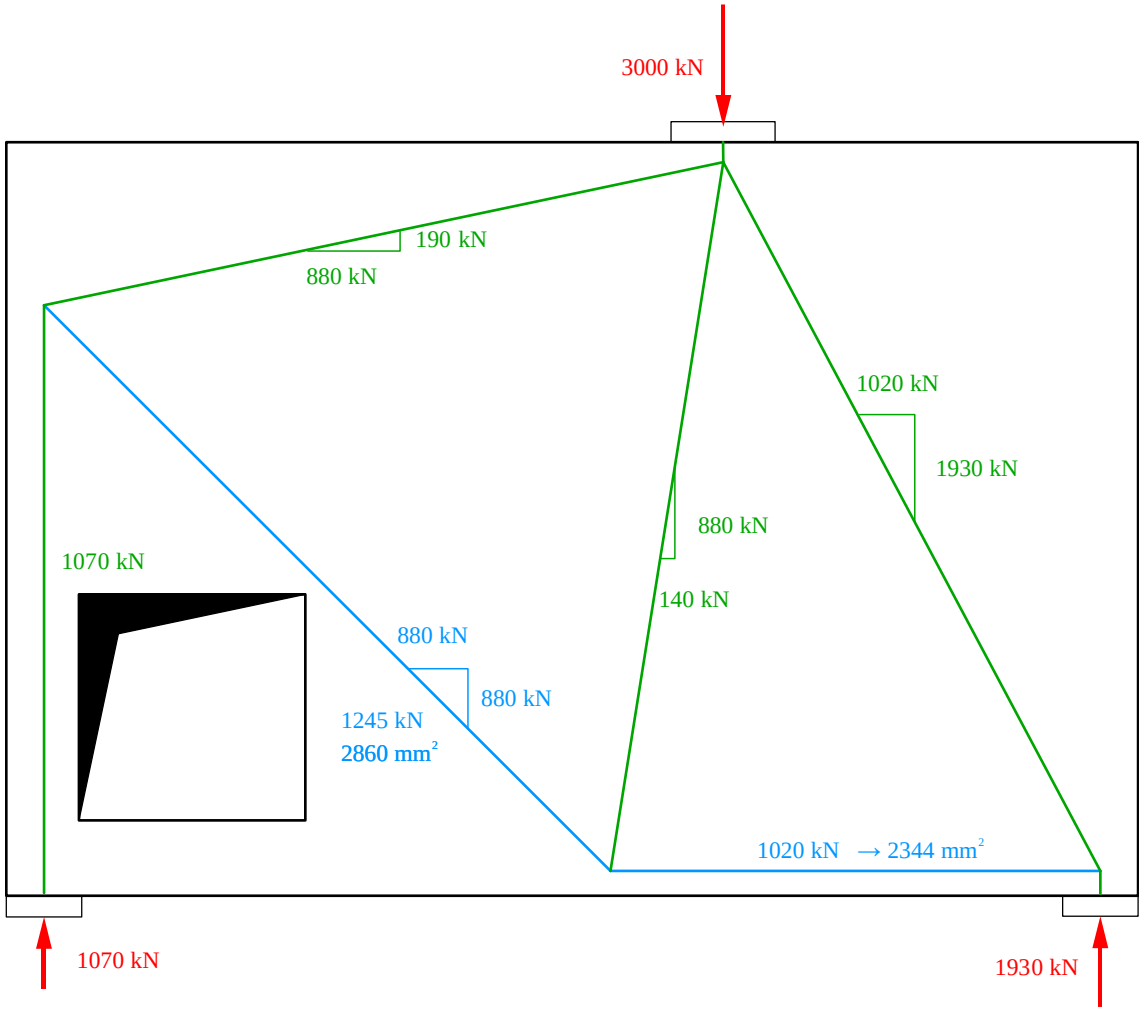


Fig. 14: Example of a strut-and tie model for Beam 2.