ETH

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 2020

Advanced Structural Concrete:

Workshop on Compatibility-based Stress Fields

(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 2.2 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 <u>complete the first two tasks before the workshop based on the guidelines provided in this document</u> (this should take approx. 30-60'). The issues and questions 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 in the following.

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/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 in: https://www.ideastatica.com/support-center
- If problem persists contact Idea StatiCa via de Live Chat

ETH

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 2020

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/all?product=concrete&category=tutorial&label=csfm

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 in the beginning of the workshop. An additional example of a dapped-end beam with an opening is documented in the following. This is a more advanced example that shows non-standard features of the program and proposed tasks. Alternatively to the online examples, the following example could be used as well, but is more time-consuming. The student should bring questions/issues to be discussed in the beginning of the workshop.

Design of dapped-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 orange framed area under **Project**, you can create a new project or open an existing project. Under **File** you can manage the units, number of significate digits, as well as other preferences (not related to the mechanical model) and the license.

Check the settings for preferences and units and adjust them to your preferences.
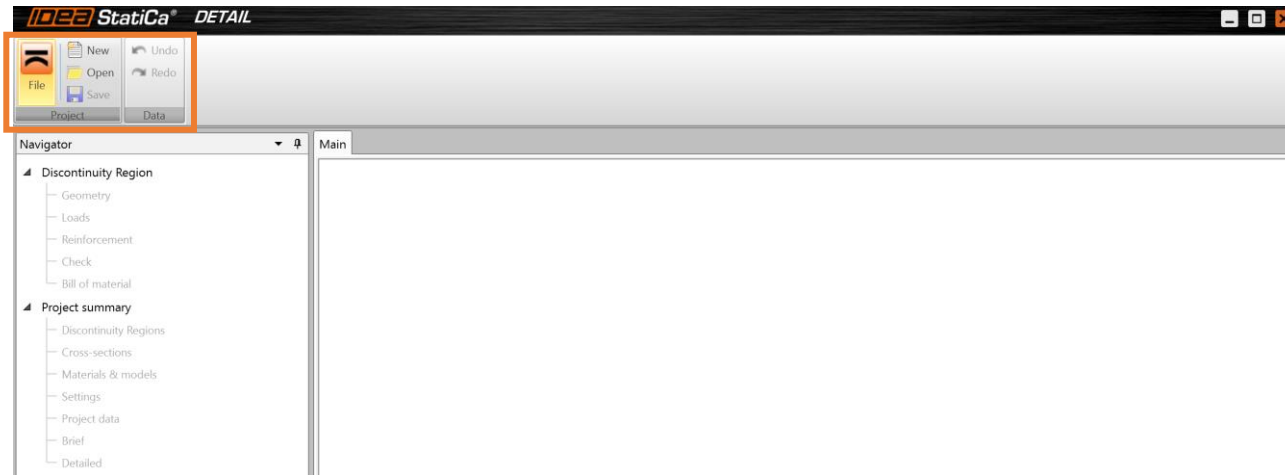

Fig. 1: User interface at startup.

Chair of Concrete Structures and Bridge Design
Institute of Structural Engineering
D-BAUG, Master's degree in Civil Engineering
Autumn Semester 2020

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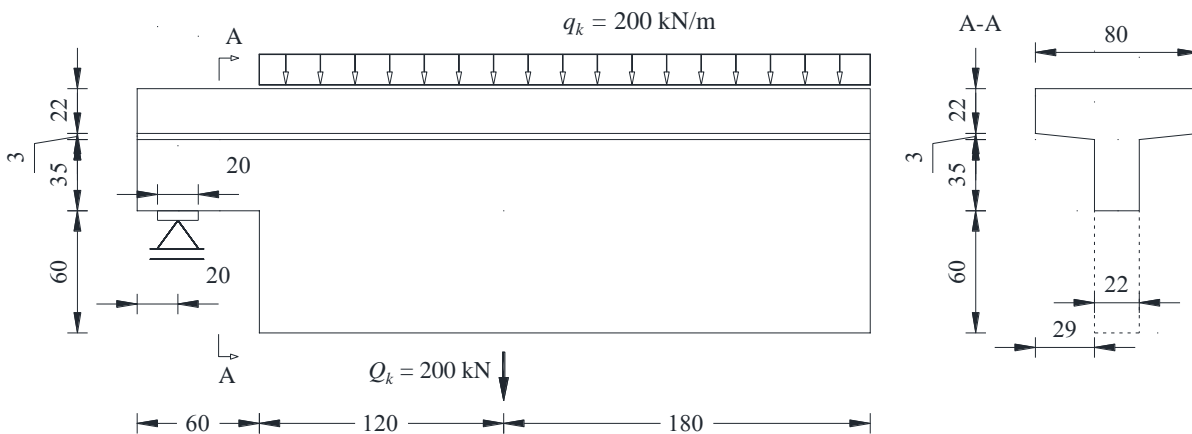
Fig. 2: Dapped-end T-beam with variable loads. Dimensions in cm.

To start building the model, a new project is opened via the icon ⬜ New . The window shown in Fig. 3a) appears. A name and author for the project should be given. Then select the **Beams** option (NB: the different templates lead to different input properties, but the resultant mechanical model is in all cases the same; it is recommended not using any template: general input). In the next window (seeFig. 3b)), you can now select the topology of the structure. Choose the **dapped-end with two cross sections**.

Go to **Project Summary** in the Navigator menu on the left → **Cross - sections** and copy the T-section. Set its height to 600 mm. Go back to **Discontinuity Region → Geometry** and modify the beam cross-sections according to the specifications in the Table 1.
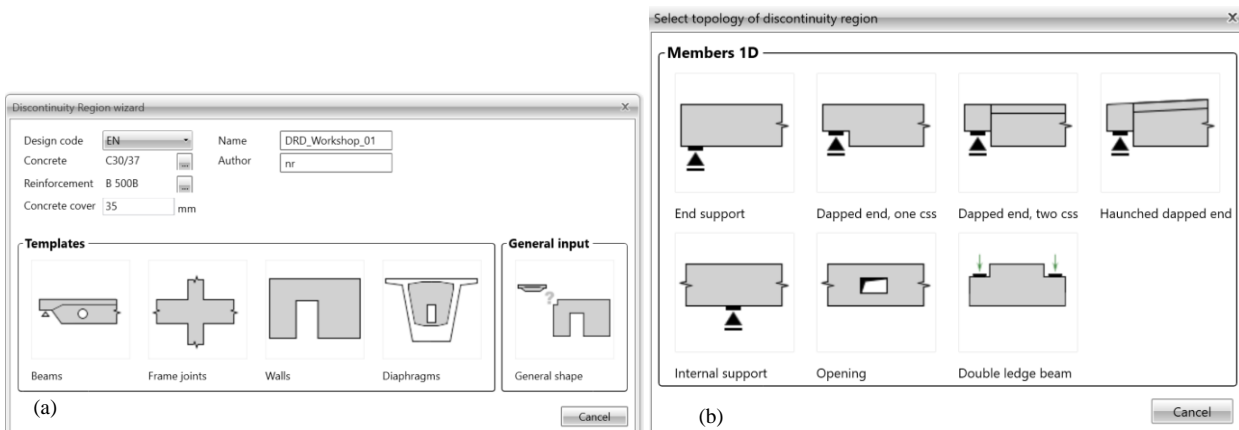

Fig. 3: (a) Discontinuity Region Wizard; (b) Select topology of discontinuity region.

Chair of Concrete Structures and Bridge Design
Institute of Structural Engineering
D-BAUG, Master's degree in Civil Engineering
Autumn Semester 2020

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

* Tip: **Concrete** C35/45 ✏ 🔄 Use the green button on the right to load a new material from the library, which then appears in the drop-down list. Existing materials can be modified manually using the pencil icon.

Using ➕ you can introduce further subregions.

Now it will be shown how to introduce the hanging punctual load by using the hanging load feature. Select the **hanging load** and in the position tab select the Masterpoint M2, the bottom edge and 1.8 m from the end of the section (see Fig. 4). Select the type **L-shaped** branches and set the dimensions to $L_2 = 1.0$ m, $L_3 = 0.40$ m and $\varnothing = 20$ mm.
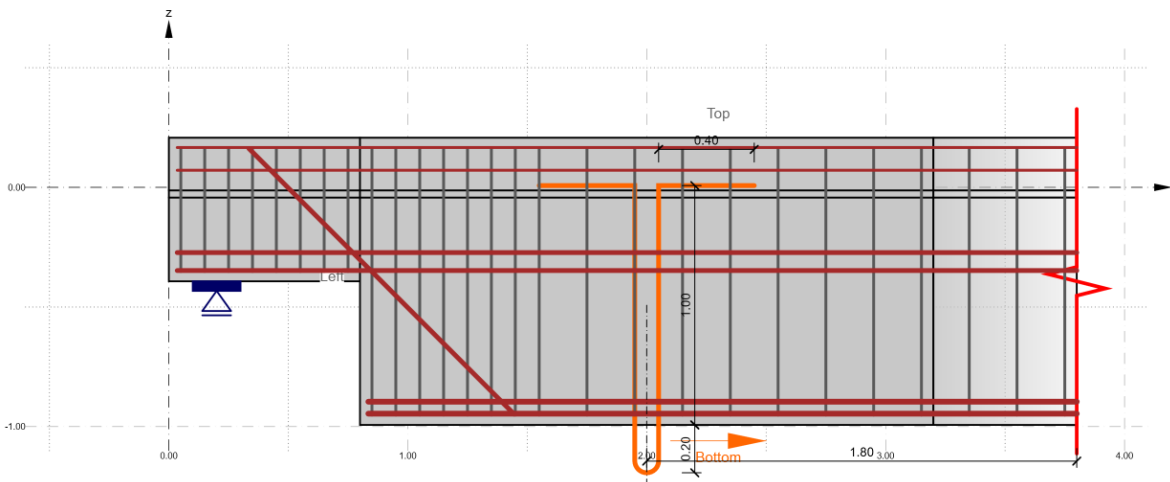

Fig. 4: Introduction of a hanging load.

Chair of Concrete Structures and Bridge Design
Institute of Structural Engineering
D-BAUG, Master's degree in Civil Engineering
Autumn Semester 2020

2.2 Loads

When creating a new project and selecting a topology, the program first goes to the **geometry** section.

In the Navigator menu, switch to the next section **Loads** to specify the loads (see Fig. 5).

NB: You can navigate freely in the navigator list not following a sequential order. For example you can jump to the **Project summary** under **Materials & models** to specify further aspects such as the material constitutive laws in more detail (this is a fundamental section). Furthermore you can find summaries of the project - the topology as a 3D-model, all used cross-sections, the material properties and their modelling (modifiable), general settings for the modelling of the reinforcement, as well as detailed compilations for reports (all these are more secondary tabs).
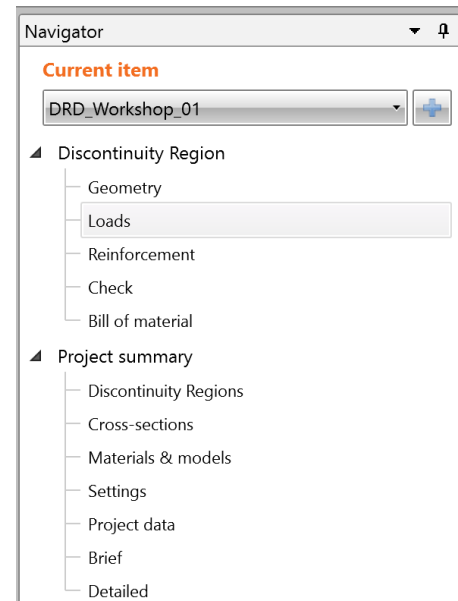

Fig. 5: Navigator.

Dead load:	Under the tab "Load Impulses" the dead loads should be introduced as line loads in the dapped-end area. Use the cross-sectional area data from Table 2 and calculate using a spatial self-load of the reinforced concrete of 25 kN/m^3.

Table 2: Cross-sectional area of the beam sections

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

Make sure that the line loads are assigned as a load case to the load combination LC1 of permanent loads type.

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

Variable load:	Under the tab "Load cases & Combinations" you can now create a load case LC2 for variable loads by clicking in the icon [Load case] . Before changing back to "Load Impulses", change the load type to "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:	Go to the tab "Internal Forces" and 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).

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 are not required (only definition of load impulses).
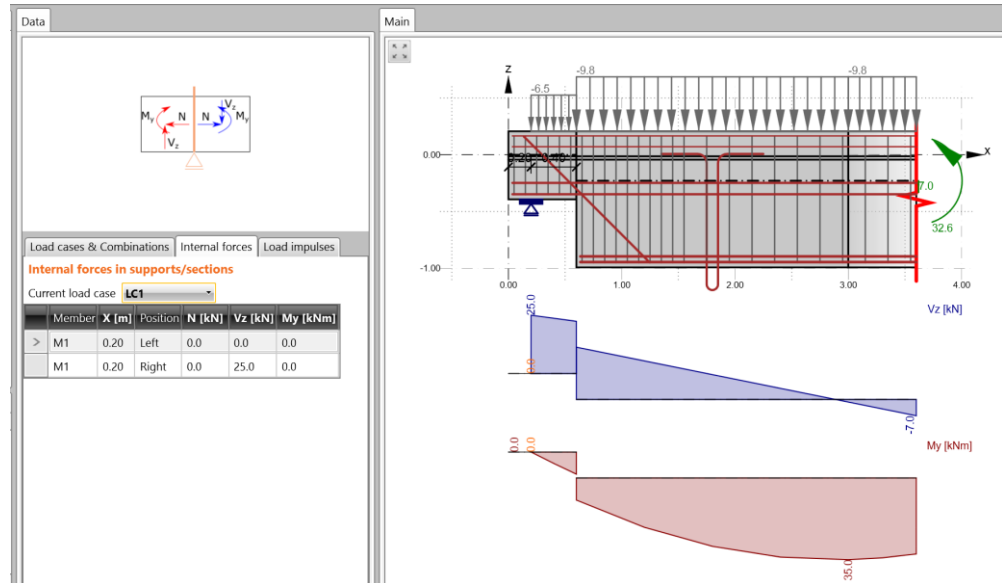
Chair of Concrete Structures and Bridge Design
Institute of Structural Engineering
D-BAUG, Master's degree in Civil Engineering
Autumn Semester 2020

Fig. 6: Internal forces for permanent loads.

**Load combina-
tion:**
Finally, the load combinations should be introduced [Combination] . In the Ultimate
Limit State (ULS), the permanent dead loads are combined with a factor of 1.35 and
the variable loads with a factor of 1.5. Create two additional load combinations for the
serviceability limit state (SLS) checks. Choose characteristic loads for one and quasi-
permanent loads for the other. Enter the weighting 1.0 for the characteristic load com-
bination and a reduction coefficient 0.3 for the variable load (1.0 for permanent loads)
for the quasi-permanent load combination.

2.3 Reinforcement

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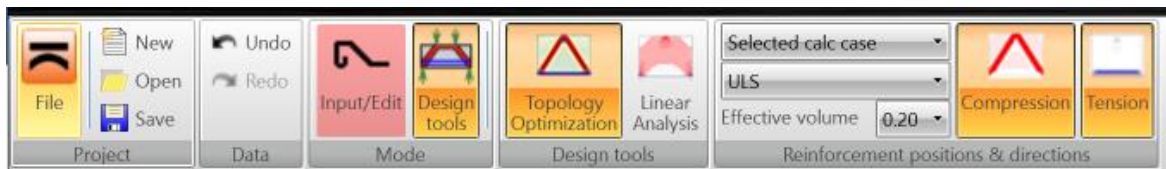areas can be defined.

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 pro-
vide a different location, adapted to the structural needs. Moreover, these tools do not provide any infor-
mation on the required reinforcement content.

Chair of Concrete Structures and Bridge Design
Institute of Structural Engineering
D-BAUG, Master's degree in Civil Engineering
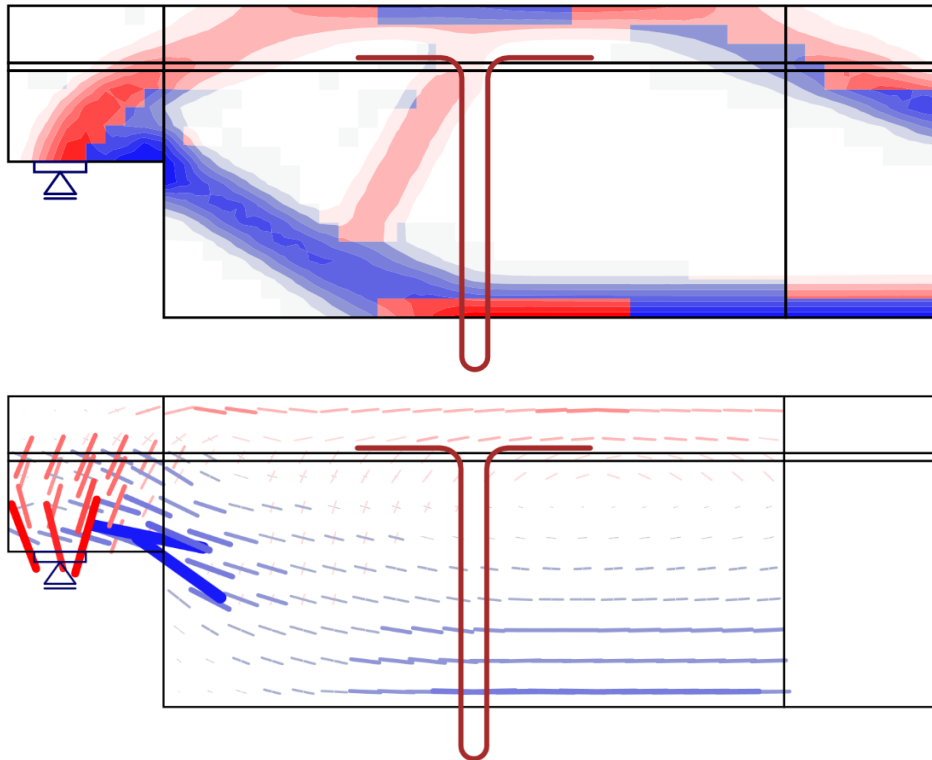Autumn Semester 2020

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 where the reinforcement could be located.
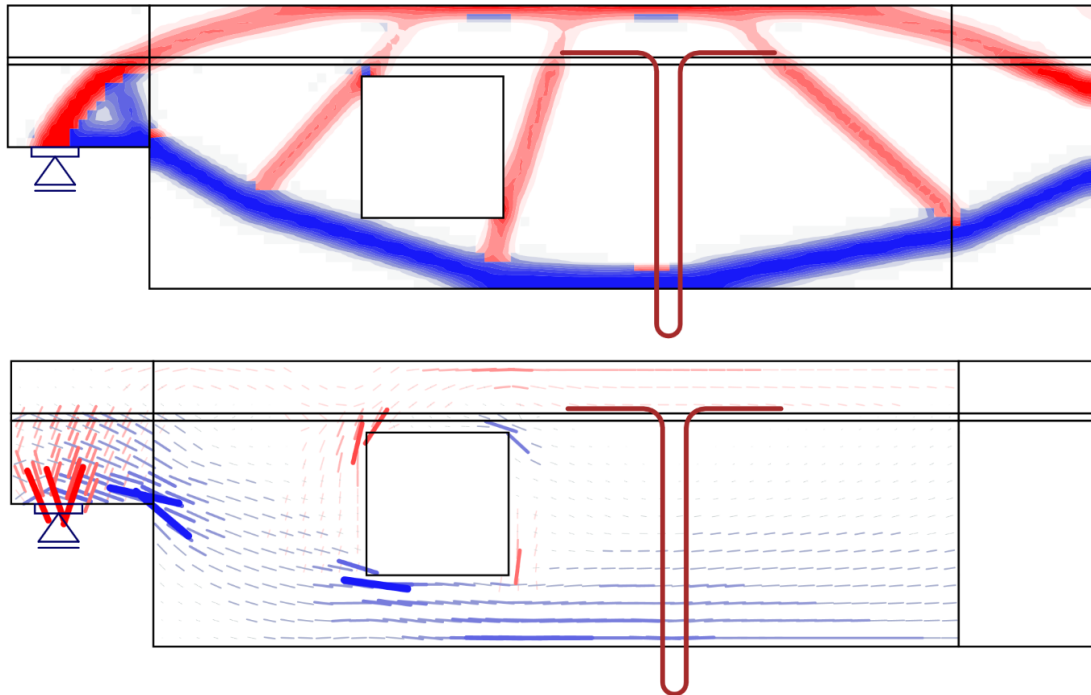
Chair of Concrete Structures and Bridge Design
Institute of Structural Engineering
D-BAUG, Master's degree in Civil Engineering
Autumn Semester 2020

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** > **Setup** 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.
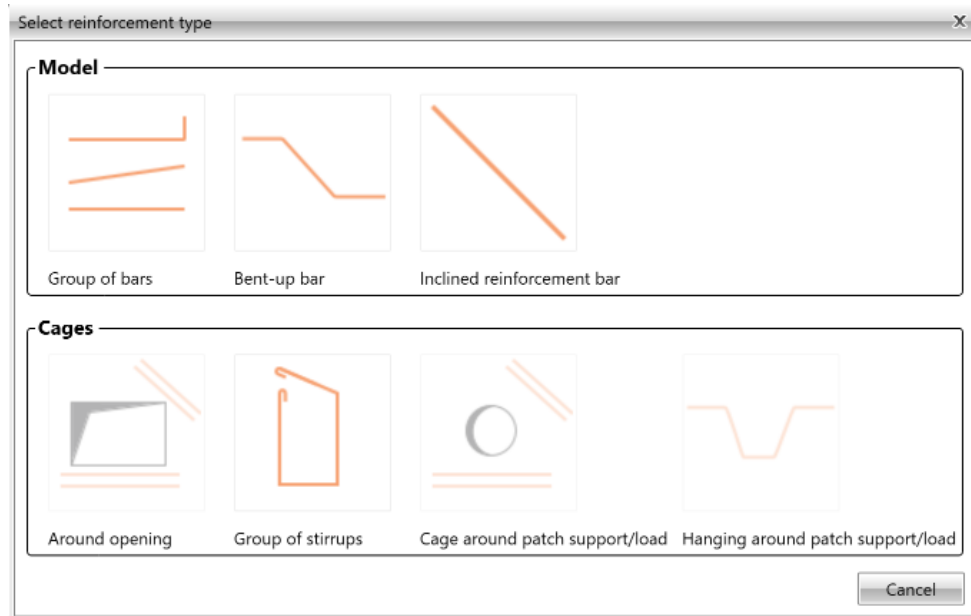
Chair of Concrete Structures and Bridge Design
Institute of Structural Engineering
D-BAUG, Master's degree in Civil Engineering
Autumn Semester 2020

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. 11b)). The 'Multiplier of flange depth' parameter can be changes by the user in the Check tab, under the Setup menu.
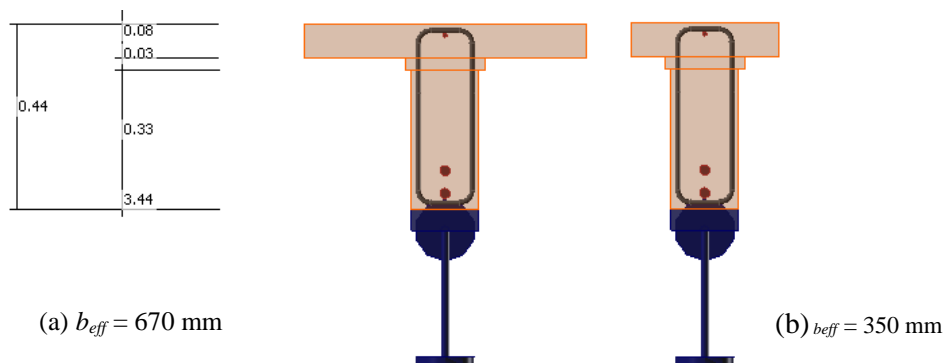

(a) $b_{eff} = 670$ mm   (b) $b_{eff} = 350$ mm

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.

Chair of Concrete Structures and Bridge Design
Institute of Structural Engineering
D-BAUG, Master's degree in Civil Engineering
Autumn Semester 2020

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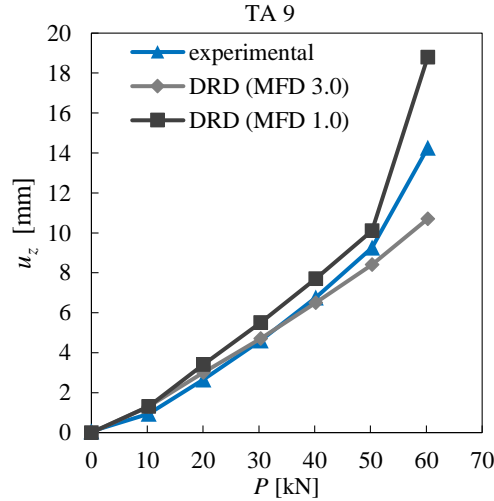

Fig. 12: Load deformation curves of the bending test TA 9 by Leonhardt & Walther (1963).

3 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.

Chair of Concrete Structures and Bridge Design
Institute of Structural Engineering
D-BAUG, Master's degree in Civil Engineering
Autumn Semester 2020

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, calculate the crack widths for a serviceability load of 60% of the given ultimate design load.

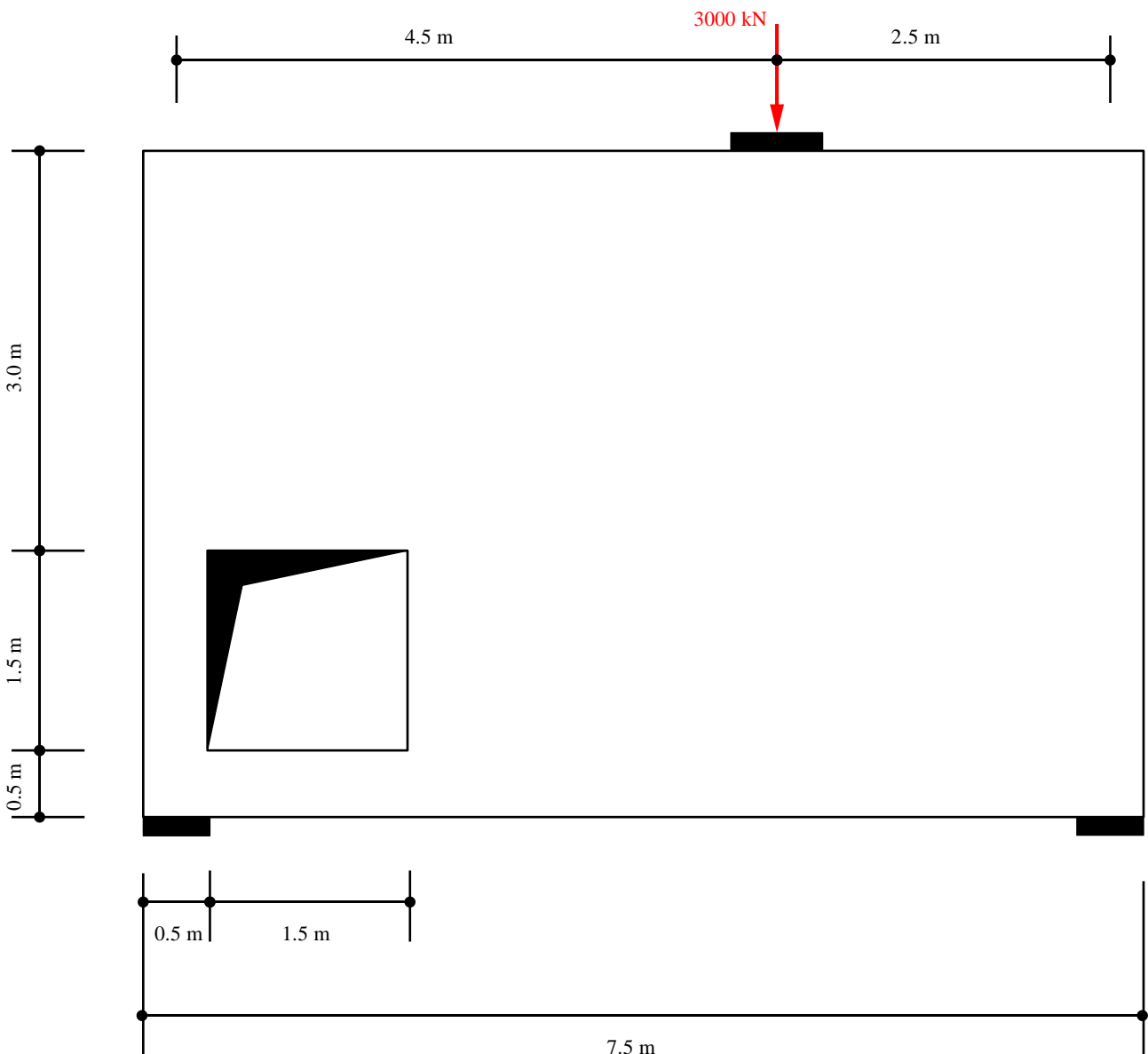Materials in all cases: concrete C30/37 and reinforcement B500B

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

Chair of Concrete Structures and Bridge Design
Institute of Structural Engineering
D-BAUG, Master's degree in Civil Engineering
Autumn Semester 2020

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.

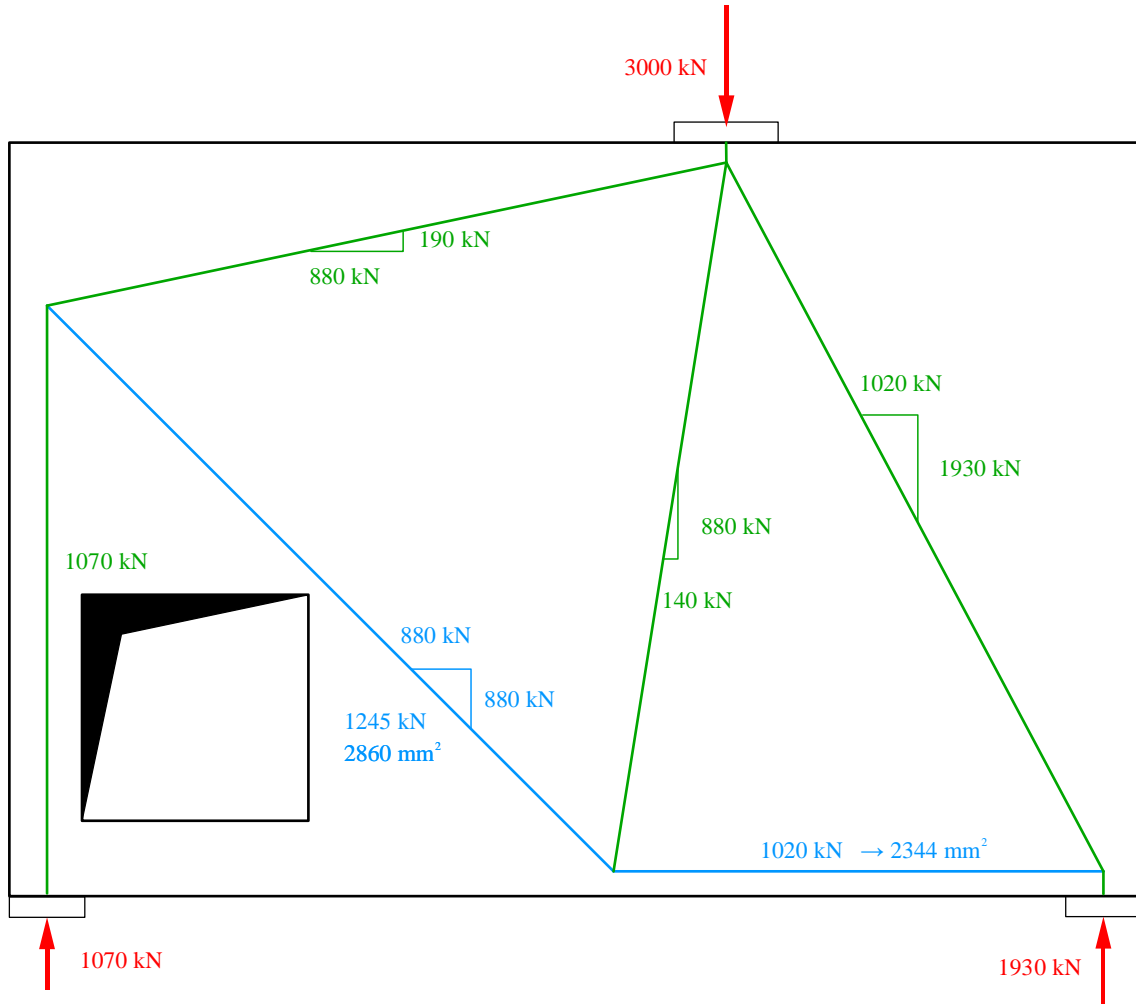In a next step, include the minimum reinforcement (recommendation: 0.2 %). What changes do you observe?

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