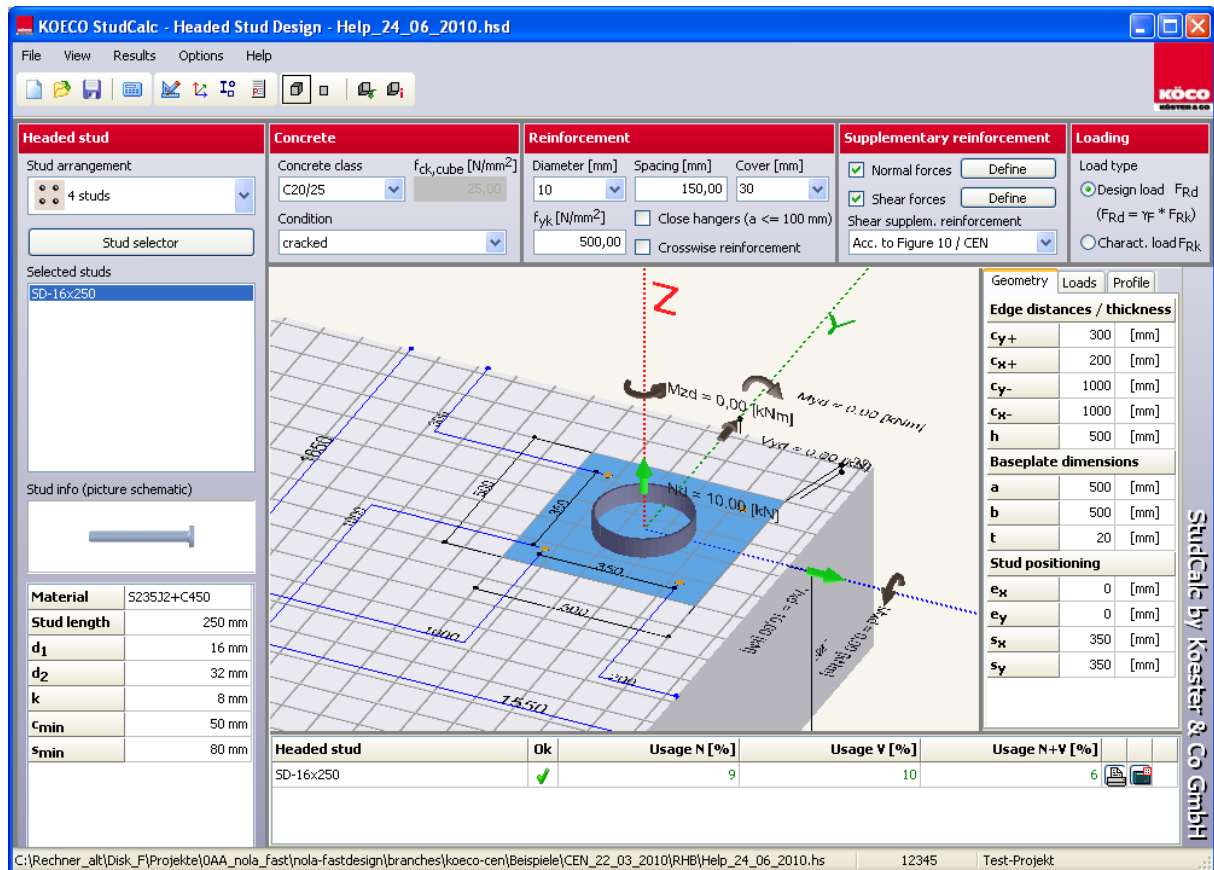




HELP MANUAL

KÖCO StudCalc – Headed Stud Design

Version 2.0

**Contact: Köster & Co.GmbH - Bolzenschweißtechnik,
Spreeler Weg 32, DE-58256 Ennepetal**

Contents

1 Introduction.....	3
1.1 Overall description.....	3
1.2 General assumptions for the calculation.....	3
2 Application setup	4
3 Graphical User Interface description.....	6
3.1 Main window.....	6
3.2 Tooltips	7
3.3 Tab Order	7
3.4 Program startup.....	7
3.5 Main Menu.....	9
3.5.1 File	9
3.5.2 View	10
3.5.3 Results	11
3.5.4 Options.....	11
3.5.5 Help.....	12
3.6 Toolbar	12
3.7 Edit View	14
3.7.1 Headed studs (stud data).....	14
3.7.2 Material data and loading (general)	15
3.7.3 Geometrical data.....	18
3.7.4 Loads	20
3.7.5 Profile data	24
3.7.6 Project and company information	26
3.7.7 3D-Object Edit View	28
3.7.8 Result Grid	29
3.8 Program dialogs	30
3.8.1 Options dialog	30
3.8.2 Disclaimer dialog.....	31
3.8.3 Graphic control information dialog	32
3.8.4 Dialogs for the definition of the supplementary reinforcement	32
3.8.5 Selection of baseplate material dialog – baseplate design.....	36
3.8.6 Scope check information dialog	36
3.9 System dialogs	37
3.9.1 Open Dialog	37
3.9.2 Save Dialog.....	38
3.9.3 Print Dialog.....	39
3.10 Database and stud selector	40
3.10.1 Stud selector	40
3.11 Reports.....	42
3.11.1 Report Preview	42

1 Introduction

1.1 Overall description

The KÖCO StudCalc – Headed Stud Design program is a project planning tool, which provides customers with all needed information about KÖCO headed studs and their design according to the CEN/TS 1992-4:2009 Part 1 and 2 and the corresponding ETA (European Technical Approval). With an easy graphical user interface the building materials (e.g. concrete and reinforcement incl. supplementary reinforcement), the whole stud arrangement and geometry (e.g. number and arrangement of studs, selection of studs,...), the geometry (e.g. edge distances, stud spacings, base plate sizes,...), the loads (e.g. characteristic or design loads, static loading,...), the profile (e.g. profile type, profile size,...) and the project data (e.g. project-No. location, part,...) can be defined.

The design of the fastening will be done by a calculation in the background of the program. The results of this calculation can be used by engineers to allow for designing the selected studs (stored in an internal database) as well as the corresponding base plate (required thickness calculated in a FE-analysis). The results will be presented in a print-preview and can be saved as a pdf-document.

1.2 General assumptions for the calculation

The KÖCO StudCalc – Headed Stud Design program is conceived for fastenings in concrete. The design of the fastenings follows the partial safety concept. The design acc. to the stud approval (ETA) and the CEN/TS 1992-4:2009 Part 2 is based on the cc-method for headed studs with characteristic values (resistances).

The studs can be used in the tensile zone (cracked concrete) or in the (evidenced) compressive zone (uncracked concrete). The program enables the design of single or multiple fastenings with edge influence (edge distances c_i smaller than 10-times effective embedment depth h_{ef}) or without edge influence ($c_i \geq h_{ef}$).

The base material has to be made of normal (or high) strength concrete. According to the CEN/TS 1992-4:2009 Part 1 and 2 the strength has to be between C20/25 up to C90/105. Smaller or higher strengths are not allowed. Internally – in the calculation – the strength will be restricted to C50/60.

The loading of the fasteners as well as of the base material is permitted for quasi-static (predominantly static) loads and for dynamic (not predominantly static) actions. The following loads can be applied (design or characteristic loads): normal force (N), shear forces (V_x , V_y) and bending moments (M_x , M_y , M_z).

The calculation of the reactions (studs and concrete) is based on the following assumptions:

- the base plate/anchor plate is rigid (Bernoulli-Hypothesis)
- tensile forces will be carried by the studs
- compressive forces will be carried by concrete and studs
- the material behaviour (bolts and concrete) is linear-elastic

- Youngs modulus concrete (30000 N/mm²)
- Youngs modulus steel 200000 N/mm²

In general the program checks (for the different possible failures) whether the design value of actions (S_d) is smaller or equal than the design value of resistance (R_d). If $S_d \leq R_d$ (utilisation $\leq 100\%$) the proof is ok.

A more detailed description about the assumptions and the calculation procedures can be found in the CEN/TS 1992-4:2009 Part 1 and 2 in combination with the approval (ETA).

Notes on the use and the design of supplementary reinforcement acc. to CEN/TS 1992-4:2009 Part 2 (see section 3.8.4):

If supplementary reinforcement was chosen by the user the corresponding concrete failure modes (tension: concrete cone failure, shear: concrete edge failure), which are obsolete due to the reinforcement will be **not** verified additionally, i. e. there will **no** decision whether a supplementary reinforcement is necessary or not internally in the program. The results – as long as possible – are optimized with respect to the amount and/or the spacing of the supplementary reinforcement. Therefore the following use of the program is suggested.

1) calculation and design of the fastening without any supplementary reinforcement. If the utilisations of the corresponding concrete failure modes (tension: concrete cone failure, shear: concrete edge failure) are $> 100\%$, the selection of a supplementary reinforcement may be reasonable.

2) selection of the corresponding supplementary reinforcement „tension“ and/or „shear“ (2 modes see section 3.8.4) as well as user definitions of reinforcement size, yield strength, etc. with a subsequent calculation and design of the fastening.

3) variation of the user definitions of supplementary reinforcement if applicable.

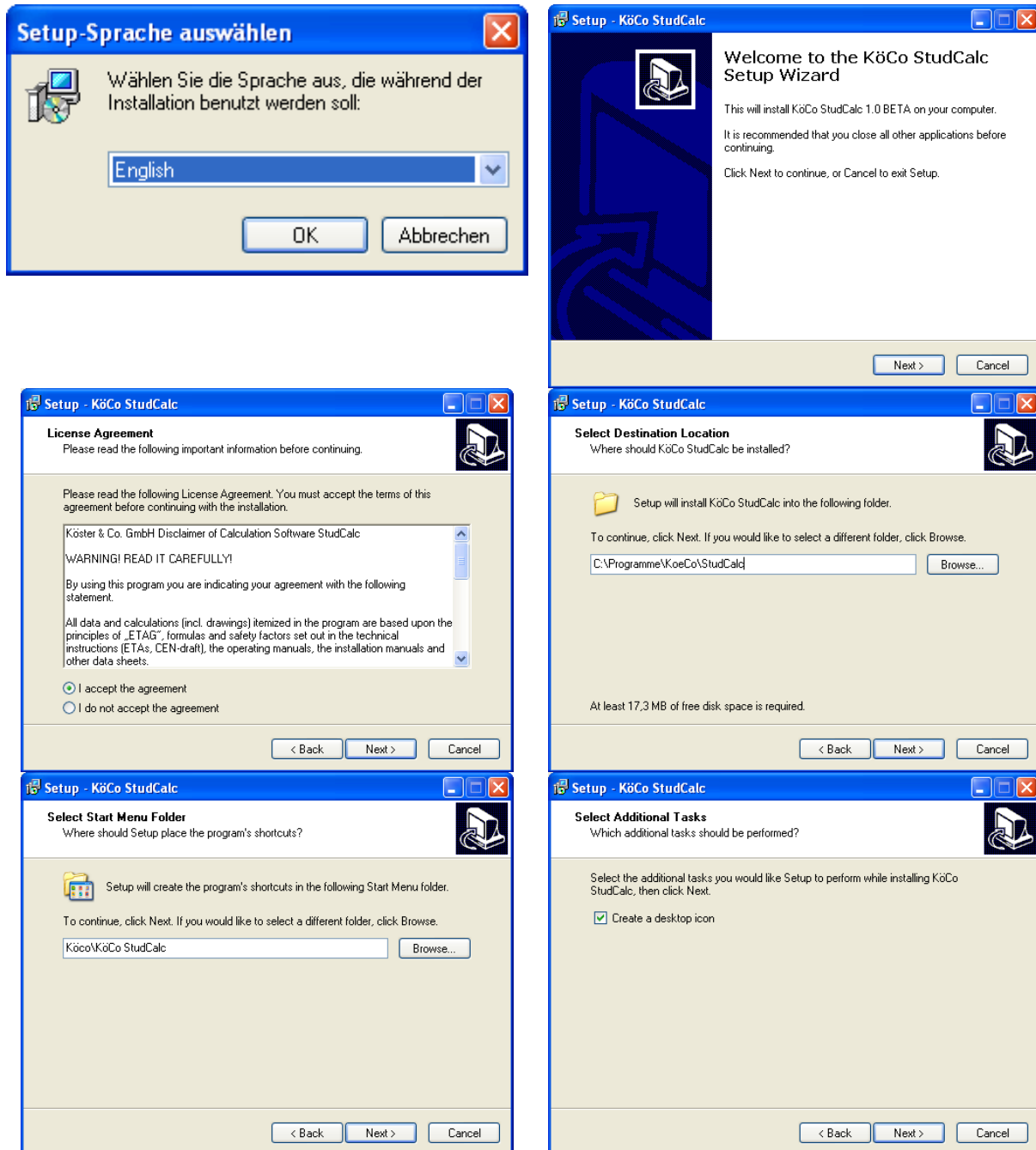
2 Application setup

The installation process will be done with the help of an external installation tool (INNO Setup). So it is guaranteed that a Windows standard installation method is proceeded. The installation process contains the following elements:

- **selection of language:** a preselection of the initial language for the installation process.
- **legal disclaimer:** a dialog containing the program disclaimer is shown. The disclaimer has to be accepted to install the program.
- **selection of program destination:** the path of the application can be chosen.
- **selection of start menu folder:** the name of the application group in the start menu can be chosen.
- **selection of additonal tasks:** create a desktop icon?
- **requirements computer/system:** Description of the minimum and recommended requirements of the computer hardware and the operating system (shown in a html-readme-file after the installation process). The

minimum requirements have to be maintained absolutely to ensure a proper functioning of the program. Furthermore a few specials about the Open GL graphics are shown.

The steps of the installation are shown in Figure 2.1.



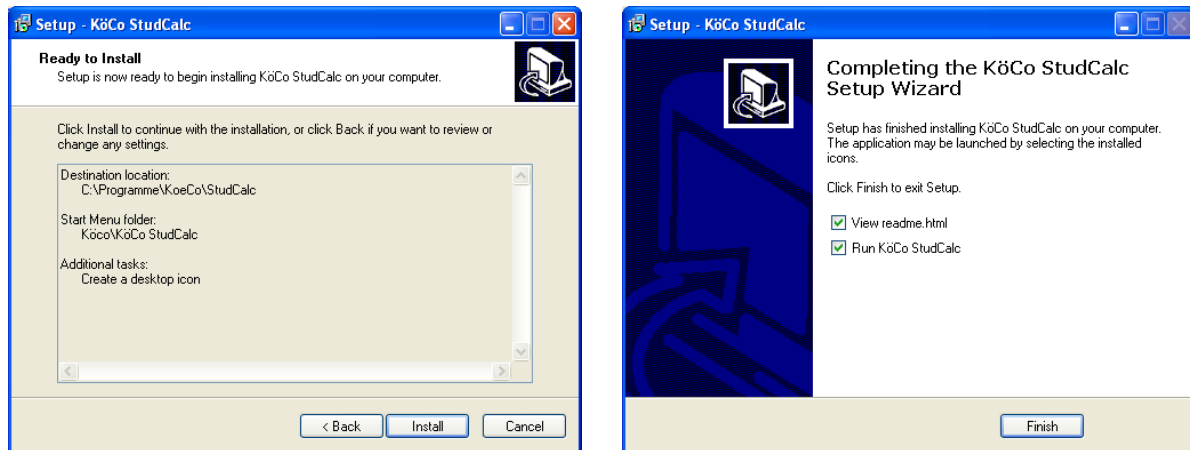


Figure 2.1: Setup process

3 Graphical User Interface description

This chapter describes the appearance of the graphical user interface of the KÖCO StudCalc – Headed Stud Design program as well as the input (see also chapter 3.10 Database) and output facilities (see chapter 3.11 Reports). For a better understanding of the discussed things screenshots are placed close to the corresponding descriptions.

3.1 Main window

Figure 3.1 shows the appearance of the KÖCO StudCalc – Headed Stud Design program *Main window*. The different components of the *Main Window* like the *Main Menu*, the *Toolbar*, the *Edit Frames* and the *3D Object Edit View* are described in detail in the following sections. Furthermore there is a *Status Bar* that shows information about the opened project (save path, project-No. and project name).

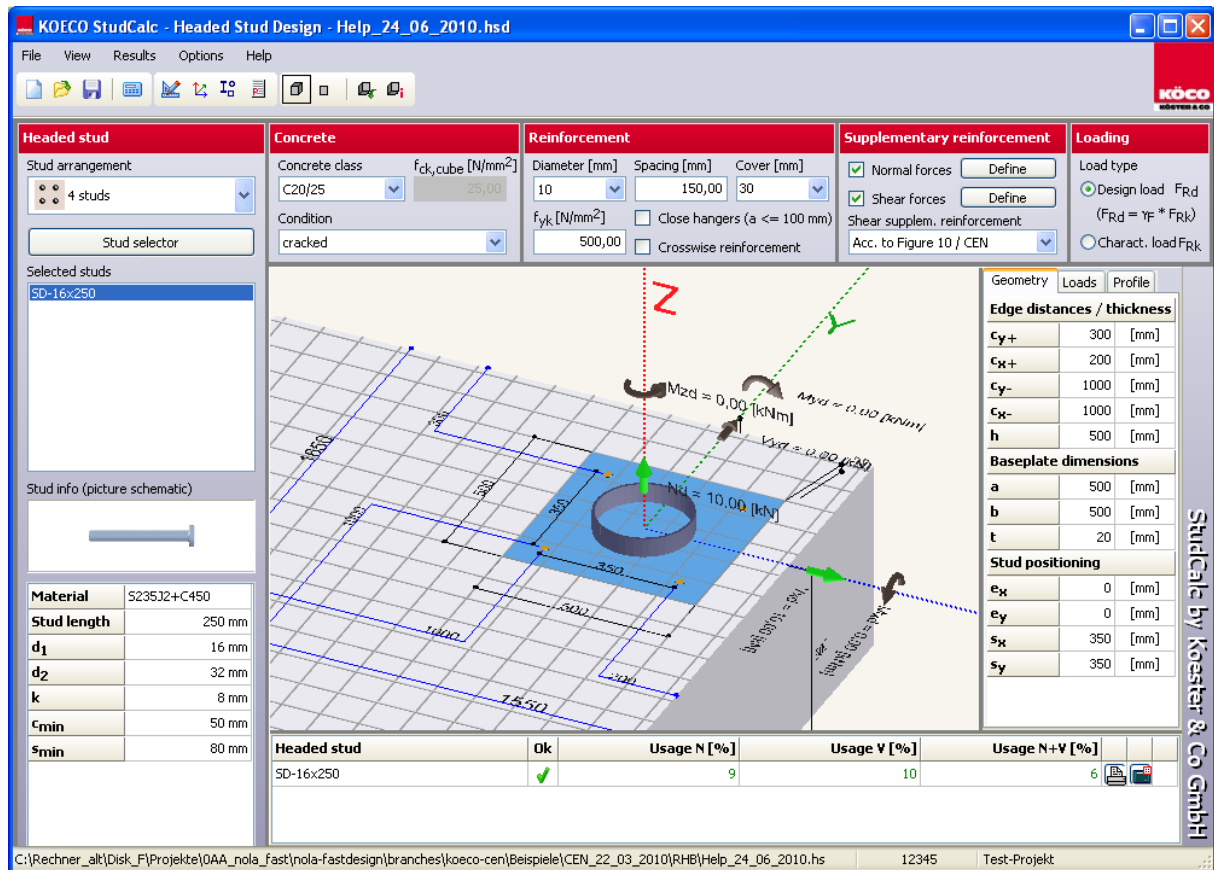


Figure 3.1: The Main Window

3.2 Tooltips

Tooltips are available almost for every visual component in the system interface. As soon as the mouse cursor is over such a component and rest there for a short while, a tooltip will appear with an appropriate description of the function of the visual component(s).

3.3 Tab Order

The tab order is defined according to the appearance of the visual components on the screen (going from top to down and from left to right). That means that a navigation through the input fields with the help of the tab key is possible. An exclusive utilisation of the tab key is not possible. Similar to the tab key, the cursor keys of the keyboard may also be used in most cases. The whole program can not be controlled with the keyboard only. For some actions a mouse is needed (e.g. for graphical manipulations (rotation, translation and zoom in/zoom out) in the *3D Object Edit View* (see chapter 3.7.7 for details).

3.4 Program startup

Immediately after the KÖCO StudCalc – Headed Stud Design program is started a *Splash-Screen* is displayed (see Figure 3.2), while the program is loaded. Before each program startup after that the disclaimer dialog (see Figure 3.3) will be shown. The mentioned conditions have to be accepted otherwise working with the program is

[illegible]

2. **New project:** If the program is started for the first time or the last session was finished without an open project a standard project with predefined (default) values is opened after the startup.
3. **Last opened project:** If a project was saved when the last session was finished that project will be opened again after the startup if the corresponding option is set in the *Options Dialog* (see chapter 3.8.1 for details). If it cannot be opened because the project file is not accessible, then a standard project with predefined (default) values will be created instead.
4. **Language:** If the program is started for the first time all program components will be in the language chosen during the application setup (see chapter 2 for details). If the program is started not for the first time all program components will be in the language chosen in the *Options Dialog* (see chapter 3.8.1 for details).
5. **Automatic online update:** If the option *Automatically search for updates* is set in the *Options Dialog* the program will connect (after start) to the KÖCO update server to check whether new updates are available. This option is not set by default.

3.5 Main Menu

The *Main Menu* for choosing actions in the KÖCO StudCalc – Headed Stud Design program is placed Windows-standard-like on top of the application frame. This section will describe the functionality of all menu items.

3.5.1 File

The *File-Menu* contains all functions concerning file operations. All actions will take effect on the *. hsd-files (**headed stud design**) which save the whole data of a project designed in the KÖCO StudCalc – Headed Stud Design program

1. **New:** When the menu item *New* is selected and since the last save procedure something was changed, a dialog box appears with the following text: "Do you want to save the current project?" and the buttons *Yes/No/Cancel*.
 1. **Yes:** The last changes since the last save procedure of the current project will be saved and a new project will be created.
 2. **No:** The last changes since the last save procedure of the current project will not be saved and a new project will be created.
 3. **Cancel:** No action will be performed.

In all cases the dialog box will disappear.

2. **Open...:** When the menu item *Open* is selected and since the last save procedure something was changed, a dialog box appears with the following text: "Do you want to save the current project?" and the buttons *Yes/No/Cancel*.

1. **Yes:** The last changes since the last save procedure of the current project will be saved and a Windows-standard Open-Dialog for opening a project will appear. See chapter 3.9.1 for details about the Open-Dialog.
2. **No:** The last changes since the last save procedure of the current project will not be saved and a Windows-standard Open-Dialog for opening a new project will appear. See chapter 3.9.1 for details about the Open-Dialog.
3. **Cancel:** No action will be performed.

In all cases the dialog box will disappear.

3. **Save:** When the menu item *Save* is selected the current project data will be saved. If the project wasn't saved before the same action as described in *Save as* will be performed.
4. **Save as...:** When the menu item *Save as* is selected a Windows-standard Save-Dialog will appear. See chapter 3.9.2 for details about the Save-Dialog.
5. **Exit:** When the menu item *Exit* is selected and since the last save procedure something was changed, a dialog box appears with the following text: "Do you want to save the current project?" and the buttons *Yes/No/Cancel*.
 1. **Yes:** The last changes since the last save procedure of the current project will be saved and the program will terminate.
 2. **No:** The last changes since the last save procedure of the current project will not be saved and the program will terminate.
 3. **Cancel:** No action will be performed.

3.5.2 View

The *View-Menu* contains all functions concerning the different input frames and the control of the graphic (different views, ...) in the *3D-Object Edit View* in a project. The different items show different opportunities for the data input.

1. **Geometry:** When the menu item *Geometry* is selected the corresponding frame will be shown. The position and the size of the frame will be saved after the program is closed and will be shown when the program is restarted accordingly. (see chapter 3.7.3)
2. **Loads:** When the menu item *Loads* is selected the corresponding frame will be shown. The position and the size of the frame will be saved after the program is closed and will be shown when the program is restarted accordingly. (see chapter 3.7.4).
3. **Profile:** When the menu item *Profile* is selected the corresponding frame will be shown. The position and the size of the frame will be saved after the

program is closed and will be shown when the program is restarted accordingly. (see chapter 3.7.5).

4. **Projectdata:** When the menu item *Projectdata* is selected the corresponding frame will be shown. The position and the size of the frame will be saved after the program is closed and will be shown when the program is restarted accordingly. (see chapter 3.7.6).
5. **3D-View:** When the menu item *3D-View* is selected the *3D Object Edit View* is in the 3-dimensional mode (full-scale).
6. **2D-View:** When the menu item *2D-View* is selected the *3D Object Edit View* is in the 2-dimensional mode (full-scale).
7. **Default-view:** When the menu item *Default-View* is selected the *3D Object Edit View* will be reset to the original start-up view (in 2D and 3D, respectively).
8. **Grafic-Info:** When the menu item *Grafic-Info* is selected a dialog with information about the control of the graphic in the *3D Object Edit View* will be shown (see chapter 3.8.3). The dialog will be closed when the button *Ok* is pressed.

3.5.3 Results

The *Results-Menu* contains one function *Calculate* which starts the calculation and after that, if the calculation is possible, the results will be displayed in the *Results Grid*. For further information about project reports and printing see chapter 3.11

3.5.4 Options

The *Options-Menu* contains all functions concerning the environment setup of the KÖCO StudCalc – Headed Stud Design program.

1. **Program options...:** When the menu item *Program options* is selected the *Options Dialog* for general things will appear. For more details see chapter 3.8.1
2. **Online update...:** When the menu item *Online update* is selected the program connects to a defined KÖCO update server and looks for available updates for the KÖCO StudCalc – Headed Stud Design program. If there is an update available, a dialog with the buttons *Update now* and *Close* will appear. If there is no update available a message “No update available at the moment” will be displayed.
 1. **Update now:** If *Update now* is selected the update will be downloaded (the update progress will be displayed) and installed. Afterwards the program will be restarted.
 2. **Close:** No action will be performed.

For more details see chapter 3.8

3.5.5 Help

The *Help-Menu* contains a few items for opening several documents or dialogs (documents (PDF) with corresponding submenu, the program disclaimer and the About dialog) which give general informations to the user.

1. **Documents (PDF):** When the menu item *Documents (PDF)* is selected or while the mouse cursor stays over it a submenu with the following entries will be shown:
Manual
ETA
When one of these submenu items is selected the corresponding file (pdf-format) will be displayed, if an appropriate program (e.g. Acrobat Reader) is installed.
2. **Disclaimer...:** When the menu item *Disclaimer...* is selected a window displaying the legal program disclaimer is opened. The dialog will be closed when the button *Ok* is pressed.
3. **About...:** When the menu item *About...* is selected a window which includes the copyright information of the KÖCO StudCalc – Headed Stud Design program and other informations is shown.

3.6 Toolbar

The *Toolbar* which is located under the *Main Menu* (see chapter 3.5 for details) is a tool to execute certain functions of the *Main Menu* faster by just clicking on the button. The functionality behind the buttons is the same as in the *Main Menu*. Figure 3.4 shows the *Toolbar Buttons*.

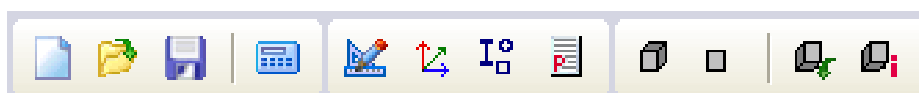


Figure 3.4: The Toolbar Buttons

Beginning from left the buttons are called:

- New Project
- Open Project
- Save Project
- Calculate
- Geometry
- Loads

- Profile
- Projectdata
- 3D-View
- 2D-View
- Default-View
- Grafic-Info

The following paragraph describes the functionality of each Toolbar Button.

1. **Toolbar Button *New Project***: See chapter 3.5.1 - *New* for the functionality of this button
2. **Toolbar Button *Open Project***: See chapter 3.5.1 - *Open* for the functionality of this button
3. **Toolbar Button *Save Project***: See chapter 3.5.1 - *Save* for the functionality of this button
4. **Toolbar Button *Calculate*** See chapter 3.5.3 - *Calculate* for the functionality of this button
5. **Toolbar Button *Geometry*** See chapter 3.5.2 - *Geometry* for the functionality of this button
6. **Toolbar Button *Loads*** See chapter 3.5.2 - *Loads* for the functionality of this button
7. **Toolbar Button *Profile*** See chapter 3.5.2 - *Profile* for the functionality of this button
8. **Toolbar Button *Projectdata*** See chapter 3.5.2 - *Projectdata* for the functionality of this button
9. **Toolbar Button *3D-View*** See chapter 3.5.2 – *3D-View* for the functionality of this button
10. **Toolbar Button *2D-View*** See chapter 3.5.2 - *2D-View* for the functionality of this button
11. **Toolbar Button *Default-View*** See chapter 3.5.2 - *Default-View* for the functionality of this button
12. **Toolbar Button *Grafic-Info*** See chapter 3.5.2 - *Grafic-Info* for the functionality of this button

3.7 Edit View

In the following section all possible input frames (docked tabs or undocked dialogs) and the functionality of the *Edit View* are described. The *Edit View* is the place for editing and showing all data concerning a project. The different views / frames can be shown by clicking a *Button* in the *Toolbar* or by selecting the corresponding menu item (see chapter 3.5.2).

3.7.1 Headed studs (stud data)

The input area *Headed stud* makes a set of input items available to define the required inputs concerning the stud data (stud arrangement). The input area *Headed stud* is divided into two sections *Stud arrangement / Stud selector* and *Selected studs* with corresponding *Stud info*. These sections contain combo boxes, buttons, edit boxes and info fields (see Figure 3.5). All input data that is defined here has an effect on the calculation process.

Headed stud

Stud arrangement

6 studs (Y-direction)

Stud selector

Selected studs

SD-13x250

Stud info (picture schematic)

Material	S235J2+C450
Stud length	250 mm
d ₁	13 mm
d ₂	25 mm
k	8 mm
h _{ef}	262 mm
c _{min}	50 mm
c _{cr,N}	393 mm
s _{min}	70 mm
s _{cr,N}	786 mm

Figure 3.5: The input area Headed stud

Section *Stud arrangement / Stud selector*

1. **Combo Box *Stud arrangement*:** The combo box *Stud arrangement* allows to choose different stud arrangements (number of studs and group type). The possible selections are: 1 stud, 2 studs (X-direction), 2 studs (Y-direction), 3 studs (X-direction), 3 studs (Y-direction), 4 studs, 6 studs (X-direction), 6 studs (Y-direction) und 9 studs.
2. **Button *Stud selector*:** When the button *Stud selector* is clicked a dialog for the selection of studs will be opened (see chapter 3.10.1 for details).

Section *Stud informations*

1. **List Box *Selected studs*:** The list box *Selected studs* contains all studs which were selected in the *Stud selector*. When a stud is selected/highlighted from the list box the corresponding stud infos (described in the following and stored in the database) will be shown.

The following informations of the selected stud (list box) are given: photo of the stud, stud material, stud length, diameter d_1 , head diameter d_2 , head thickness k , effective embedment depth h_{ef} , minimum edge distance c_{min} , characteristic edge distance $c_{cr,N}$, minimum stud spacing s_{min} , characteristic stud spacing $s_{cr,N}$.

3.7.2 Material data and loading (general)

Input area Concrete

The input area *Concrete* makes a set of input items available to define the required inputs concerning the concrete material properties. The input area *Concrete* contains combo boxes and an edit box (see Figure 3.6). All input data that is defined here has an effect on the calculation process.

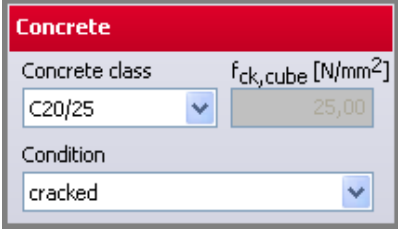


Figure 3.6: The input area Concrete

1. **Combo Box *Concrete class*:** The combo box *Concrete class* allows to choose the concrete compressive strength (concrete class C – cylinder / cube compressive strength) from different classes. These are the classes which are allowed for the use according to the ETAs: C20/25, C25/30, C30/37, C35/45, C40/50, C45/55, C50/60. Furthermore *user-defined* can be selected.
2. **Combo Box *Condition*:** The combo box *Condition* allows to choose the concrete state: uncracked or cracked.

3. **Edit Box $f_{ck,cube}$:** In the edit box $f_{ck,cube}$ values of the concrete cube compressive strength can be inputted if *user-defined* is selected in the combo box *Concrete class*. Otherwise the edit box is set to read-only.

Input area Reinforcement

The input area *Reinforcement* makes a set of input items available to define the required inputs concerning the reinforcement in the concrete member. The input area *Reinforcement* contains combo boxes, edit boxes and check boxes (see Figure 3.7). All input data that is defined here has an effect on the calculation process.

Figure 3.7: The input area Reinforcement

1. **Combo Box *Diameter*:** The combo box *Diameter* allows to choose different diameters of the longitudinal reinforcement. The following selections are possible: 0, 4, 6, 8, 10, >= 12 mm.
2. **Edit Box *Spacing*:** In the edit box *Spacing* the spacing/distance of the reinforcing bars (longitudinal reinforcement) can be set.
3. **Combo Edit Box *Cover*:** The combo edit box *Cover* allows to choose or to input the different concrete covers of the longitudinal reinforcement. The following selections are possible: 15, 20, 25, 30, 35, 40, 45, 50, 55 mm.
4. **Edit Box f_{yk} :** In the edit box f_{yk} the characteristic yield strength of the reinforcing bars (longitudinal reinforcement) can be set in N/mm².
5. **Check Box *Close hangers (a <= 100 mm)*:** With the Check-Box *Close hangers (a <= 100 mm)* it can be defined whether a corresponding edge reinforcement is present or not.
6. **Check Box *Crosswise reinforcement*:** With the Check-Box *Crosswise reinforcement* it can be defined whether the longitudinal reinforcement is arranged crosswise or not.

Input area Supplementary reinforcement

The input area *Supplementary reinforcement* makes a set of input items available to define an eventually present supplementary reinforcement in the area of the fastening. This input area contains combo boxes, check boxes and buttons (see Figure 3.8). All input data that is defined here has an effect on the calculation process.

Figure 3.8: The input area Supplementary reinforcement

1. **Check Box *Normal forces*:** The check box *Normal forces* defines whether the supplementary reinforcement for normal forces is present or not.
2. **Button *Define*:** This button is active only if the check box *Normal forces* is checked. When the button *Define* is clicked a dialog for the definition of the supplementary reinforcement will be opened (see chapter 3.8.4).
3. **Check Box *Shear forces*:** The check box *Shear forces* defines whether the supplementary reinforcement for shear forces is present or not.
4. **Button *Define*:** This button is active only if the check box *Shear forces* is checked. When the button *Define* is clicked a dialog for the definition of the supplementary reinforcement will be opened (see chapter 3.8.4).
5. **Combo Box *Shear suppl. reinforcement*:** The combo box *Shear suppl. reinforcement* is active only if the check box *Shear forces* is checked. The combo box allows to choose different possibilities (arrangements) of the supplementary reinforcement for shear forces. The following selections are possible: Acc. to Figure 9 / CEN and Acc. to Figure 10 / CEN.

Input area Loading

The input area *Loading* allows to define the used loadtype via a radio group. (see Figure 3.9). All input data that is defined here has an effect on the calculation process.

Figure 3.9: The input area Loading

1. **Radio Group *Load type*:** The Radio Group *Load type* allows to define the use of characteristic (F_{Rk}) or design loads ($F_{Rd} = \gamma_F * F_{Rk}$). Note: In case of charact. loads F_{Rk} the given loads will be multiplied with the corresponding partial load safety factors γ_F in the calculation (see chapter 3.7.4).

3.7.3 Geometrical data

The frame *Geometry* makes a set of input items available to define the required inputs concerning the geometry. The frame contains three sections: *Edge distances / thickness*, *Baseplate dimensions* and *Stud positioning*. Each section consists of different edit boxes (see Figure 3.10). All input data that is defined here can have an effect on the calculation process. Note: the values which are given (or changed) in the edit boxes have an influence on the *3D Object Edit View* and vice versa.

a)

Geometry		
Edge distances / thickness		
c_{y+}	300	[mm]
c_{x+}	200	[mm]
c_{y-}	1000	[mm]
c_{x-}	1000	[mm]
h	500	[mm]
Baseplate dimensions		
a	500	[mm]
b	500	[mm]
t	20	[mm]
Stud positioning		
e_x	0	[mm]
e_y	0	[mm]
s_x	350	[mm]
s_y	350	[mm]

b)

Geometry		
Edge distances / thickness		
c_{y+}	300	[mm]
c_{x+}	200	[mm]
c_{y-}	1000	[mm]
c_{x-}	1000	[mm]
h	500	[mm]
Baseplate dimensions		
a	500	[mm]
b	500	[mm]
t	20	[mm]
Stud positioning		
e_x	0	[mm]
e_y	0	[mm]
s_x	350	[mm]
s_y	350	[mm]

StudCalc by Köster & Co GmbH

345 Test-Projekt

Figure 3.10: The frame *Geometry*: a) undocked und b) docked

Section *Edge distances / thickness*

1. **Edit Box c_{y+} :** The edit box c_{y+} allows to edit the edge distance of the relevant stud (depending on the stud arrangement) in positive y-direction.
2. **Edit Box c_{x+} :** The edit box c_{x+} allows to edit the edge distance of the relevant stud (depending on the stud arrangement) in positive x-direction.
3. **Edit Box c_{y-} :** The edit box c_{y-} allows to edit the edge distance of the relevant stud (depending on the stud arrangement) in negative y-direction.
4. **Edit Box c_{x-} :** The edit box c_{x-} allows to edit the edge distance of the relevant stud (depending on the stud arrangement) in negative x-direction.
5. **Edit Box h :** The edit box h allows to edit the thickness of the concrete plate (z-direction).

Section *Baseplate dimensions*

1. **Edit Box a :** The edit box a allows to edit the width of the metal baseplate in x-direction.
2. **Edit Box b :** The edit box b allows to edit the width of the metal baseplate in y-direction.
3. **Edit Box t :** The edit box t allows to edit the thickness of the metal baseplate (z-direction).

Section *Stud positioning*

1. **Edit Box e_x :** The edit box e_x allows to edit the eccentricity of the stud group in x-direction with respect to the center of gravity of the metal baseplate.
2. **Edit Box e_y :** The edit box e_y allows to edit the eccentricity of the stud group in y-direction with respect to the center of gravity of the metal baseplate.
3. **Edit Box s_x :** The edit box s_x allows to edit the stud spacing in x-direction. Note: the availability / editability of this edit box depends on selected stud arrangement.
4. **Edit Box s_y :** The edit box s_y allows to edit the stud spacing in y-direction. Note: the availability / editability of this edit box depends on selected stud arrangement.

3.7.4 Loads

The frame *Loads* makes a set of input items available to define the required inputs concerning the loading of the construction. If the *Load type* is *Charact. load* (see above) the frame contains two sections: *Charact. loads (static)* und *Dynamic loads (% of Q)*. If the *Load type* is *Design load* (see above) the frame contains only the section *Design loads (static)*. These sections consist of different edit boxes (see

Figure 3.11). All input data that is defined here can have an effect on the calculation process. Note: the values which are given (or changed) in the combo/edit boxes have an influence on the *3D Object Edit View* and vice versa.

a1)

Loads		
Charact. loads (static)		
N_G	10,00	[kN]
V_{xG}	3,00	[kN]
V_{yG}	0,00	[kN]
M_{xG}	0,00	[kNm]
M_{yG}	5,00	[kNm]
M_{zG}	0,00	[kNm]
γ_G	1,35	
N_Q	5,00	[kN]
V_{xQ}	2,00	[kN]
V_{yQ}	0,00	[kN]
M_{xQ}	0,00	[kNm]
M_{yQ}	0,00	[kNm]
M_{zQ}	0,00	[kNm]
γ_Q	1,50	
Dynamic loads (% of Q)		
Percent	20,00	[%]

b1)

Geometry Loads Profile Projec		
Charact. loads (static)		
N_G	10,00	[kN]
V_{xG}	3,00	[kN]
V_{yG}	0,00	[kN]
M_{xG}	0,00	[kNm]
M_{yG}	5,00	[kNm]
M_{zG}	0,00	[kNm]
γ_G	1,35	
N_Q	5,00	[kN]
V_{xQ}	2,00	[kN]
V_{yQ}	0,00	[kN]
M_{xQ}	0,00	[kNm]
M_{yQ}	0,00	[kNm]
M_{zQ}	0,00	[kNm]
γ_Q	1,50	
Dynamic loads (% of Q)		
Percent	20,00	[%]

StüdelCale by Köster & Co GmbH

345 Test-Projekt

a2)

Loads		
Design loads (static)		
N_D	21,00	[kN]
V_{xD}	7,05	[kN]
V_{yD}	0,00	[kN]
M_{xD}	0,00	[kNm]
M_{yD}	6,75	[kNm]
M_{zD}	0,00	[kNm]

b2)

Geometry	Loads	Profile	Projec	<	>
Design loads (static)					
N_D	21,00	[kN]			
V_{xD}	7,05	[kN]			
V_{yD}	0,00	[kN]			
M_{xD}	0,00	[kNm]			
M_{yD}	6,75	[kNm]			
M_{zD}	0,00	[kNm]			
StudCalc by Köster & Co GmbH					
345	Test-Projekt				

Figure 3.11: The frame *Loads*: a1) Charact. load – undocked und b1) Charact. load – docked; a2) Design load – undocked und b2) Design load – docked

Sections *Charact. loads (static)* and *Design loads (static)*, respectively

1. **Edit Box N_G** : The edit box N_G allows to edit the permanent normal force (z-direction). The edit box is available if *Charact. load* is selected in the Radio Group *Load type*.
2. **Edit Box V_{xG}** : The edit box V_{xG} allows to edit the permanent shear force (x-direction). The edit box is available if *Charact. load* is selected in the Radio Group *Load type*.
3. **Edit Box V_{yG}** : The edit box V_{yG} allows to edit the permanent shear force (y-direction). The edit box is available if *Charact. load* is selected in the Radio Group *Load type*.
4. **Edit Box M_{xG}** : The edit box M_{xG} allows to edit the permanent bending moment (x-direction). The edit box is available if *Charact. load* is selected in the Radio Group *Load type*.
5. **Edit Box M_{yG}** : The edit box M_{yG} allows to edit the permanent bending moment (y-direction). The edit box is available if *Charact. load* is selected in the Radio Group *Load type*.
6. **Edit Box M_{zG}** : The edit box M_{zG} allows to edit the permanent torque moment (z-direction). The edit box is available if *Charact. load* is selected in the Radio Group *Load type*.
7. **Edit Box γ_G** : The edit box γ_G allows to edit the partial load safety factor for permanent loads. The edit box is available if *Charact. load* is selected in the Radio Group *Load type* and editable if the corresponding check box in the options dialog is set (see chapter 3.8.1 for details).
8. **Edit Box N_Q** : The edit box N_Q allows to edit the variable normal force (z-direction). The edit box is available if *Charact. load* is selected in the Radio Group *Load type*.
9. **Edit Box V_{xQ}** : The edit box V_{xQ} allows to edit the variable shear force (x-direction). The edit box is available if *Charact. load* is selected in the Radio Group *Load type*.
10. **Edit Box V_{yQ}** : The edit box V_{yQ} allows to edit the variable shear force (y-direction). The edit box is available if *Charact. load* is selected in the Radio Group *Load type*.
11. **Edit Box M_{xQ}** : The edit box M_{xQ} allows to edit the variable bending moment (x-direction). The edit box is available if *Charact. load* is selected in the Radio Group *Load type*.
12. **Edit Box M_{yQ}** : The edit box M_{yQ} allows to edit the variable bending moment (y-direction). The edit box is available if *Charact. load* is selected in the Radio Group *Load type*.

13. **Edit Box M_{zQ} :** The edit box M_{zQ} allows to edit the variable torque moment (z-direction). The edit box is available if *Charact. load* is selected in the Radio Group *Load type*.
14. **Edit Box γ_Q :** The edit box γ_Q allows to edit the partial load safety factor for variable loads. The edit box is available if *Charact. load* is selected in the Radio Group *Load type* and editable if the corresponding check box in the options dialog is set (see chapter 3.8.1 for details).
 - A. **Edit Box N_d :** The edit box N_d allows to edit the design normal force (z-direction). The edit box is available if *Design load* is selected in the Radio Group *Load type*.
 - B. **Edit Box V_{xd} :** The edit box V_{xd} allows to edit the design shear force (x-direction). The edit box is available if *Design load* is selected in the Radio Group *Load type*.
 - C. **Edit Box V_{yd} :** The edit box V_{yd} allows to edit the design shear force (y-direction). The edit box is available if *Design load* is selected in the Radio Group *Load type*.
 - D. **Edit Box M_{xd} :** The edit box M_{xd} allows to edit the design bending moment (x-direction). The edit box is available if *Design load* is selected in the Radio Group *Load type*.
 - E. **Edit Box M_{yd} :** The edit box M_{yd} allows to edit the design bending moment (y-direction). The edit box is available if *Design load* is selected in the Radio Group *Load type*.
 - F. **Edit Box M_{zd} :** The edit box M_{zd} allows to edit the design torque moment (z-direction). The edit box is available if *Design load* is selected in the Radio Group *Load type*.

Section *Dynamic loads (% of Q)*

1. **Edit Box *Percent*:** The edit box *Percent* allows to edit the percentage of dynamic loads as a part of the variable loads. Note: this load part will be used for the calculation of stress ranges. The edit box is available if *Charact. load* is selected in the Radio Group *Load type*.

3.7.5 Profile data

The frame *Profile* makes a set of input items available to define the required inputs concerning the profile. The frame contains different combo and edit boxes (see Figure 3.12). All input data that is defined here can have an effect on the calculation process. Note: the values which are given (or changed) in the combo/edit boxes have an influence on the *3D Object Edit View*.

a)

Profile		
Profile type	HEB-Profile	
Size	IPB 100 / HEB 100	
b	100	[mm]
h	100	[mm]
t	10	[mm]
$e_{p,x}$	0	[mm]
$e_{p,y}$	0	[mm]
Angle	0	[°]

b)

Profile		
Profile type	HEB-Profile	
Size	IPB 100 / HEB 100	
b	100	[mm]
h	100	[mm]
t	10	[mm]
$e_{p,x}$	0	[mm]
$e_{p,y}$	0	[mm]
Angle	0	[°]

StuCalc by Köster & Co GmbH

345 Test-Projekt

Figure 3.12: The frame *Profile*: a) undocked und b) docked

1. **Combo Box *Profile type*:** The combo box *Profile type* allows to select different types of profiles which are stored in the database.
2. **Combo Box *Size*:** The combo box *Size* allows to select different sizes of profiles which are stored in the database. The possible selections depend on the selected profile in the combo box *Profile type*.

3. **Info Box b / Width:** In the read-only info box b the width of the selected profile is displayed.
4. **Info Box h / Height:** In the read-only info box h the height of the selected profile is displayed.
5. **Info Box t / Thickness:** In the read-only info box t the thickness of the selected profile is displayed.
6. **Edit Box $e_{p,x}$:** The edit box $e_{p,x}$ allows to edit the eccentricity between the center of gravity of the profile and the center of gravity of baseplate in x-direction.
7. **Edit Box $e_{p,y}$:** The edit box $e_{p,y}$ allows to edit the eccentricity between the center of gravity of the profile and the center of gravity of baseplate in y-direction.
3. **Edit Box *Angle*:** The edit box *Angle* allows to edit the rotation of the profile (anti-clockwise-direction around longitudinal profile axis).

3.7.6 Project and company information

The frame *Project* makes a set of input items available to define administration data for a project. The *Project* frame consists of edit/memo boxes that are divided into certain sections (see Figure 3.13). All input data that is defined here hasn't any effect on the calculation process. The *Project* input is only used in the printed reports. See chapter 3.11 for details about the reports.

Section *Company data*

1. **Edit Box *Company*:** The edit box *Company* allows to input the company name.
2. **Edit Box *Contact person*:** The edit box *Contact person* allows to input the name of the contact person.
3. **Memo-Edit Box *Address*:** The edit box *Address* allows to input the address.
4. **Edit Box *Phone*:** The edit box *Phone* allows to input the phone number.
5. **Edit Box *Fax*:** The edit box *Fax* allows to input the fax number.
6. **Edit Box *email*:** The edit box *email* allows to input the email address.

Section *Project data*

1. **Combined Edit Box *Date*:** The edit box *Date* allows to input the date (also via calendar).
2. **Edit Box *Project-No.*:** The edit box *Project-No.* allows to input a project number

3. **Edit Box *Project*:** The edit box *Project* allows to input the name of a project
 4. **Edit Box *Part*:** The edit box *Part* allows to input the part of a project
 5. **Memo-Edit Box *Notes*:** The memo-edit box *Notes* allows to input several lines of notes corresponding to the project
- a)

Project and company information	
Company data	
Company	
Contact person	
Address	
Phone	
Fax	
email	
Project data	
Date	30.08.2007
Project-No.	12345
Project	Test-Projekt
Part	Part XYZ
Notes	Test notes

b)

Project and company information	
Company data	
Company	
Contact person	
Address	
Phone	
Fax	
email	
Project data	
Date	30.08.2007
Project-No.	12345
Project	Test-Projekt
Part	Part XYZ
Notes	Test notes

StuCalc by Köster & Co GmbH

345 Test-Projekt

Figure 3.13: The frame *Project and company information*: a) undocked und b) docked

3.7.7 3D-Object Edit View

Figure 3.14 shows an example how the input interface for the *3D-Object Edit View* looks like. The construction (studs, baseplate, profile, base material,...) is visualized as a 3D-object with OpenGL. The *3D-Object Edit View* is full-scale. For working with the mouse in the *3D-Object Edit View* the following mouse actions are available:

1. **Rotate:** To rotate the left mouse button has to be pressed and the mouse has to be moved in any direction.
2. **Move/Translate:** To move the shift key and the left mouse button has to be pressed and the mouse has to be moved in any direction.
3. **Zoom In:** To zoom in the right mouse button has to be pressed and the mouse has to be moved upwards. Alternatively the mouse wheel has to be "wheeled upwards".
4. **Zoom Out:** To zoom out the right mouse button has to be pressed and the mouse has to be moved downwards. Alternatively the mouse wheel has to be "wheeled downwards".
5. **Direction of arrows:** If *Design load* is selected in the Radio Group *Load type* the direction of the arrows (loads, moments) will change their direction and sign on click (left mouse button). Note: as long as the load / moment is zero the arrows are "grey". If there is a load / moment set the arrows are "green". If there are dynamic load / moment parts set the arrows are "red".

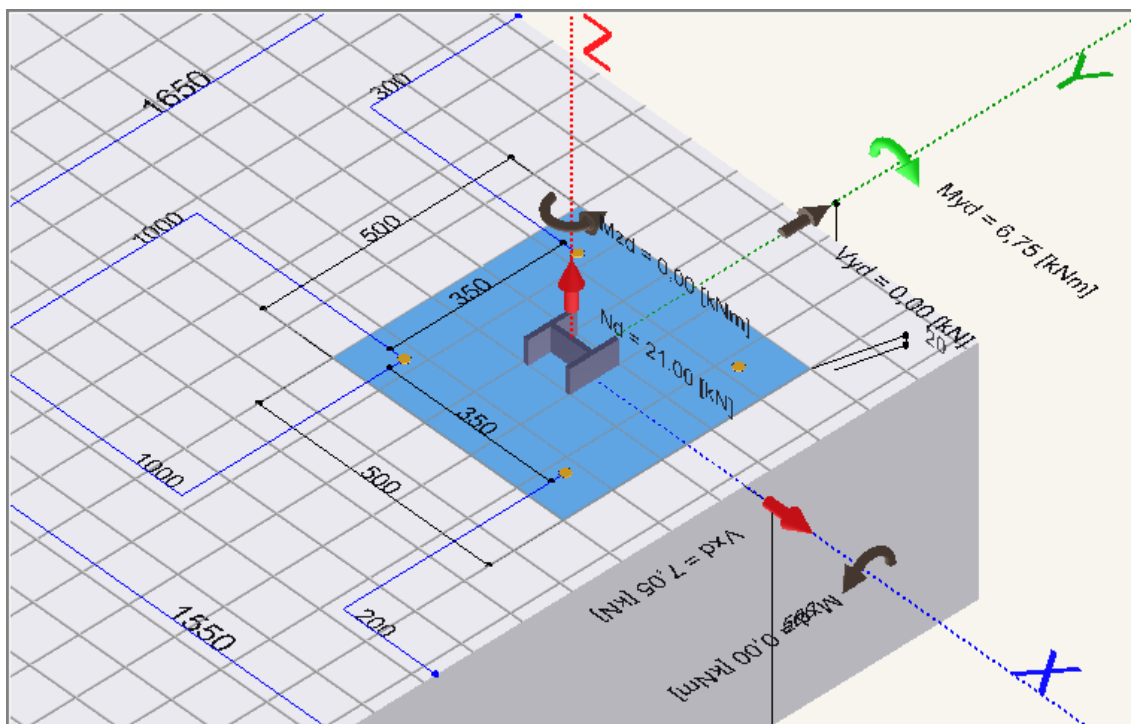


Figure 3.14: The 3D-Object Edit View

Fields which are editable (not everything what is given in the input frames!) in the 3D-

Object Edit View will be highlighted (red) when the mouse cursor is over them and the mouse cursor will change to *HandPoint*. On click (left mouse button) an Edit Box will open and the value can be edited. Note: the maximum input value is set to 10000 (mm for length values and kN/kNm for load values, respectively)

3.7.8 Result Grid

Figure 3.15 shows examples how the *Result Grid* can look like. The result grid shows either, (a) if the calculation was possible, two buttons *Report preview* and *Baseplate design* and the maximum utilisations of the proofs (normal force, shear force, interaction; Note: in case of no verification/proof is needed "na" (= not applicable) will be given in the corresponding column; if for one verification/proof no resistance can be calculated "9999" will be given in the corresponding column) or (b) one button *Scope check information*, if some boundary conditions are not fulfilled. From the *Result Grid* the following actions are available:

1. **Button *Report preview*:** When the button *Report preview* (in one cell) is clicked the *Print preview* filled with results of the corresponding stud shown in the Headed stud cell will be opened (see chapter 3.11.1 for print preview details).
2. **Button *Baseplate design*:** When the button *Baseplate design* (in one cell) is clicked a dialog for selecting the baseplate material (depending on the stud material) will be opened. If this dialog is closed by clicking on the button *Ok* the design of the baseplate (FE-analysis) will be started automatically (with or without showing the procession, see chapter 3.8.1). After that the calculated thickness of the baseplate of the corresponding stud is shown.
3. **Button *Scope check information*:** When the button *Scope check information* (in one cell) is clicked the corresponding dialog filled with informations about which scope check (boundary condition of the corresponding stud) and why is not fulfilled will be opened (see chapter 3.8.6 for details).

a)

Headed stud	Ok	Usage N [%]	Usage V [%]	Usage N+V [%]		
SD-16x400	✓	45	37	16		

b)

Headed stud	Ok	Usage N [%]	Usage V [%]	Usage N+V [%]		
SD-16x400	⚠	-	-	-		

Figure 3.15: The Result Grid

3.8 Program dialogs

In the following chapter the functions of several program dialogs that are used in the KÖCO StudCalc program are described.

3.8.1 Options dialog

The *Program Options* dialog allows to set up certain environment options for the program. It is divided into three sections *General*, *Language* und *Online update*. The contents of these sections will be described in the following. Figure 3.16 shows the dialog *Program Options*.

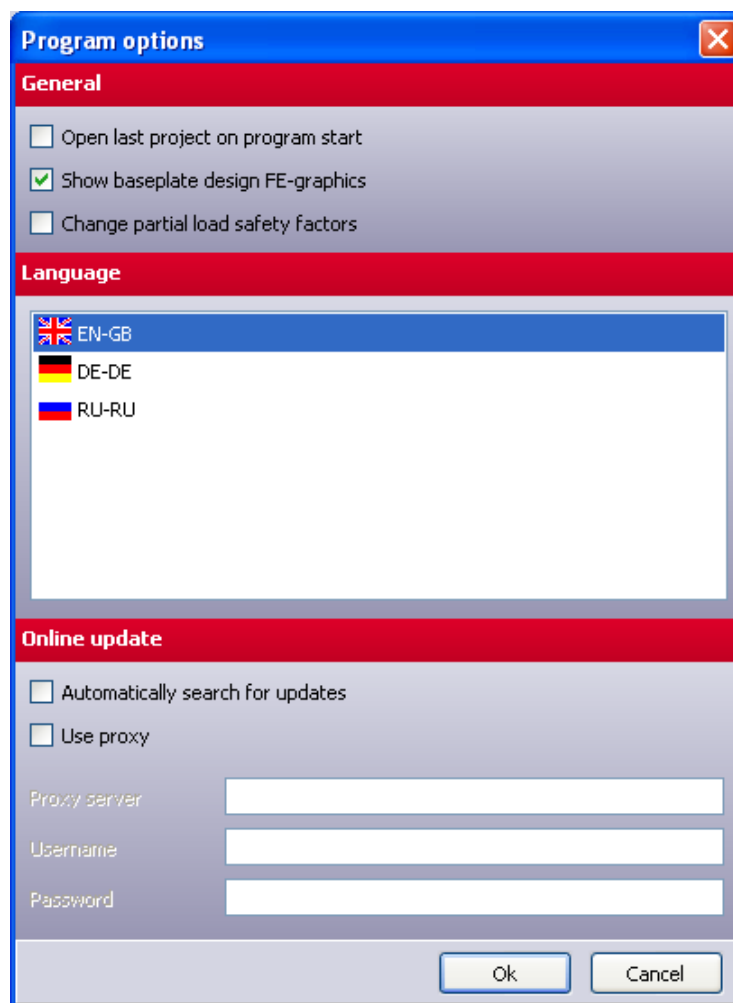


Figure 3.16: The Program options dialog

Section *General*

1. **Check Box *Open last project at program start*:** If the check box is checked then the project which was opened (and saved) when the program was closed will be opened automatically again.
2. **Check Box *Show baseplate design FE-Graphics*:** If the check box is checked then the procession of the baseplate design (FE-analysis) will be shown.

3. **Check Box *Change partial load safety factors***: If the check box is checked then the partial load safety factors in the frame *Loads* can be changed/edited.

Section *Language*

The section *Language* allows to choose the language in which the program should be displayed. Only one selection can be made within the list box.

- **Entry *EN-GB***: If the entry EN-GB is selected all program components will be labelled in "British English".
- **Entry *DE-DE***: If the entry DE-DE is selected all program components will be labelled in "German".
- **Entry *RU-RU***: If the entry RU-RU is selected all program components will be labelled in "Russian".

Section *Online update*

1. **Check Box *Automatically search for updates***: If the check box is checked then the program will automatically look for available updates (on a KÖCO update server) every time the program is started.
2. **Check Box *Use proxy***: If the check box is checked then the settings for a proxy server defined in the edit boxes described below, are used to connect to the online update site.
3. **Edit Box *Proxy Server***: Only available if *Use Proxy* is checked. Domain-name or ip-address of a proxy server.
4. **Edit Box *Username***: Only available if *Use Proxy* is checked. If the proxy server needs an username and a password to authenticate, the username has to be filled in the edit box.
5. **Edit Box *Password***: Only available if *Use Proxy* is checked. If the proxy server needs an username and a password to authenticate, the password has to be filled in the edit box.

3.8.2 Disclaimer dialog

The Disclaimer dialog (see Figure 3.17) shows the legal program disclaimer.

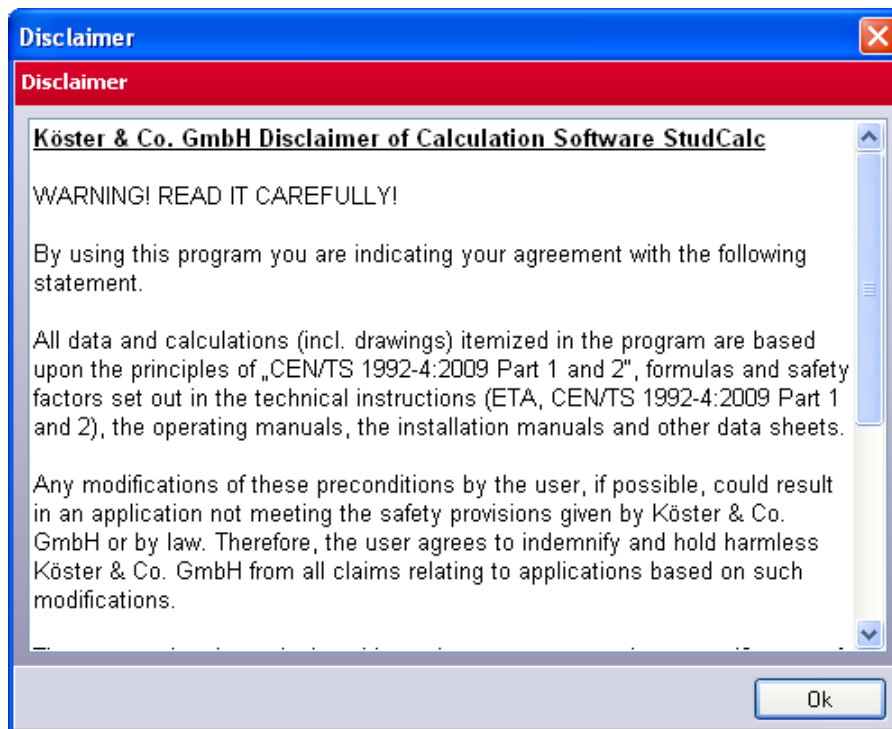


Figure 3.17: The Disclaimer dialog

3.8.3 Graphic control information dialog

The Graphic control information dialog (see Figure 3.18) shows informations about the control of the graphic in the *3D-Object Edit View*.

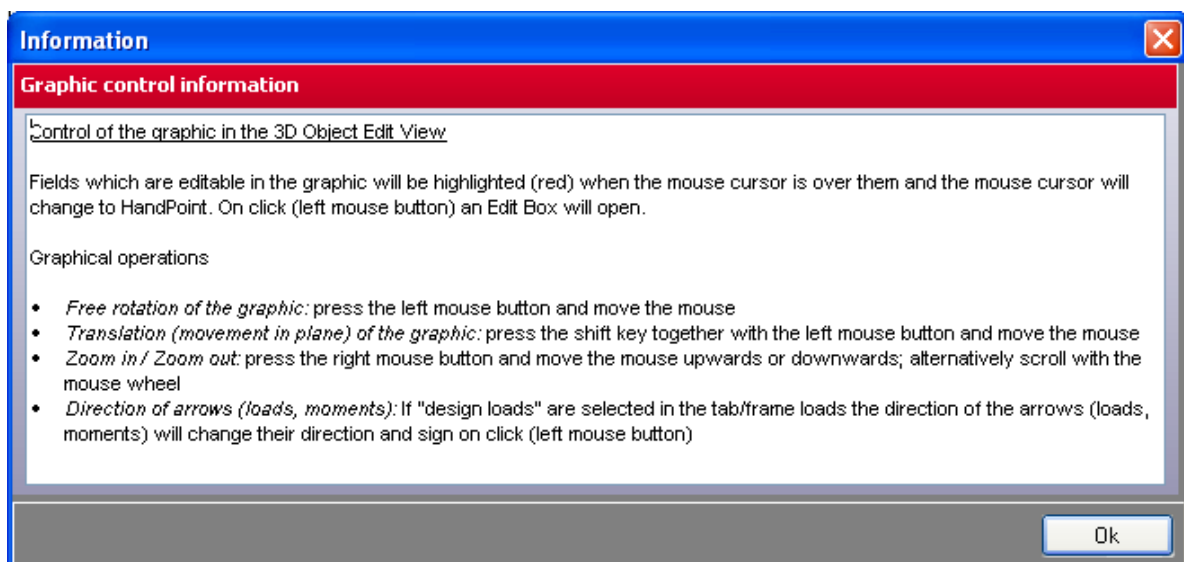


Figure 3.18: The Graphic control information dialog

3.8.4 Dialogs for the definition of the supplementary reinforcement

The *Supplementary Reinforcement* dialogs (see Figure 3.19 and Figure 3.20) contain different Radio Groups, Check Boxes and Edit Boxes (see also chapter 3.7.2). With these input fields the user is able to define the arrangement (boundary

conditions) of the supplementary reinforcement. Furthermore explaining texts and schematic drawings are presented.

General

1. **Button *Ok*:** If this button is clicked the user input data from the dialogs will be used in the program and the dialog will be closed.
2. **Button *Cancel*:** If this button is clicked no user input data from the dialogs (or the data which was already there when the dialog is opened) will be used in the program and the dialog will be closed.

Normal forces (Tension)

Supplementary reinforcement „CEN/TS 1992-4:2009 Part 2“

1. **Radio Group *Diameter*:** The Radio Group *Diameter* allows to define the diameter of the supplementary reinforcement.
2. **Edit Box f_{yk} :** The edit box f_{yk} allows to input the characteristic yield strength of the supplementary reinforcement in N/mm².
3. **Check Box *Good bond condition*:** If the check box is checked then *Good bond condition* (acc. to EC 2) is assumed in the program.
4. **Check Box *Hooked bars*:** If the check box is checked then *Hooked bars* and the corresponding values (acc. to EC 2) are assumed in the program.

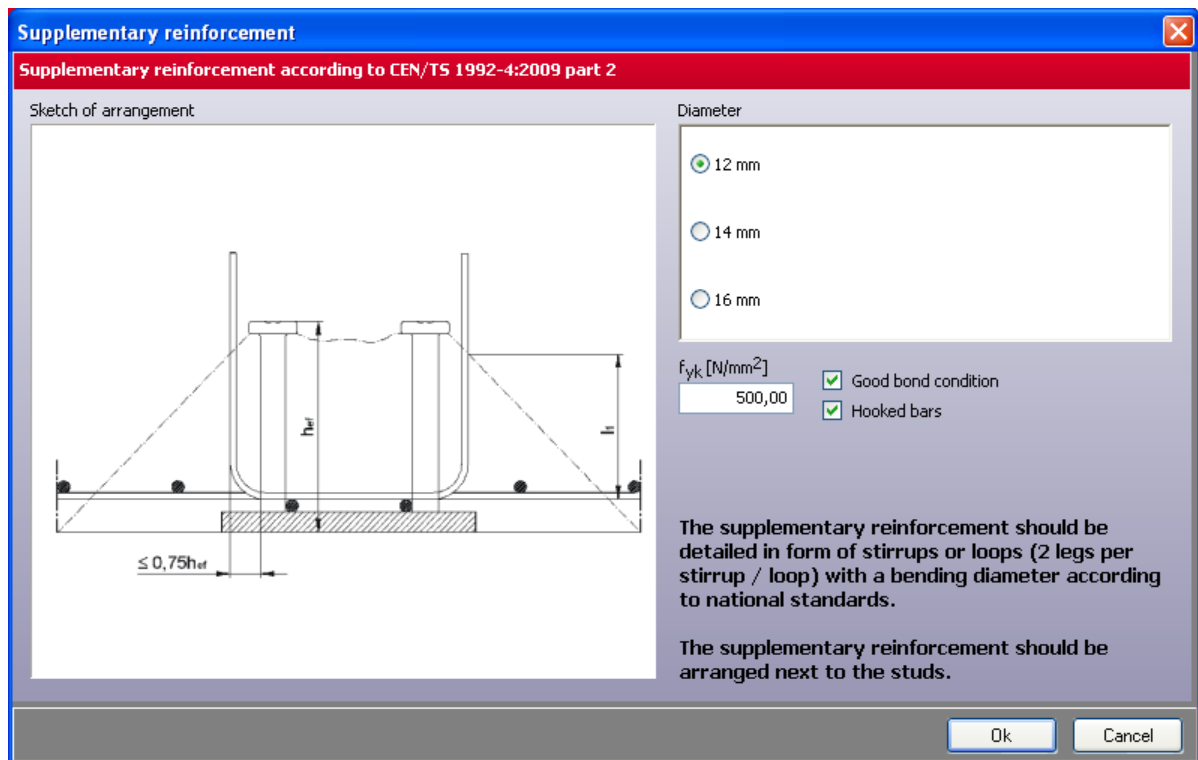


Figure 3.19: Supplementary reinforcement dialog – Normal forces (Tension): CEN/TS 1992-4:2009 Part 2

Shear forces

Supplementary reinforcement „Acc.to Figure 9 of CEN/TS 1992-4:2009 Part 2“

1. **Radio Group *Diameter*:** The Radio Group *Diameter* allows to define the diameter of the supplementary reinforcement.
2. **Edit Box f_{yk} :** The edit box f_{yk} allows to input the characteristic yield strength of the supplementary reinforcement in N/mm².
3. **Check Box *Good bond condition*:** If the check box is checked then *Good bond condition* (acc. to EC 2) is assumed in the program.
4. **Check Box *Hooked bars*:** If the check box is checked then *Hooked bars* and the corresponding values (acc. to EC 2) are assumed in the program.

Supplementary reinforcement „Acc.to Figure 10 of CEN/TS 1992-4:2009 Part 2“

1. **Radio Group *Diameter*:** The Radio Group *Diameter* allows to define the diameter of the supplementary reinforcement.
2. **Radio Group *Arrangement*:** The Radio Group *Arrangement* allows to define the arrangement of the supplementary reinforcement (number of reinforcement legs per stud).

a)

Supplementary reinforcement

Supplementary reinforcement according to CEN/TS 1992-4:2009 part 2

Sketch of arrangement

Diameter

☐ 8 mm

☐ 10 mm

☒ 12 mm

☐ 14 mm

☐ 16 mm

f_{yk} [N/mm²]

500,00

☒ Good bond condition

☒ Hooked bars

The supplementary reinforcement is in the form of a surface reinforcement according to Figure 9 (sketch) consisting of straight or hooked bars.

The supplementary reinforcement should be arranged as shown in Figure 9 (sketch).

Ok Cancel

b)

Supplementary reinforcement

Supplementary reinforcement according to CEN/TS 1992-4:2009 part 2

Sketch of arrangement

Diameter

☐ 8 mm

☐ 10 mm

☒ 12 mm

☐ 14 mm

☐ 16 mm

☒  

The supplementary reinforcement should be detailed in form of stirrups or loops (2 legs per stirrup / loop) with a bending diameter according to national standards.

The supplementary reinforcement should be arranged next to the studs.

Ok Cancel

Figure 3.20: Supplementary reinforcement dialogs – Shear forces: a) Acc. to Figure 9 of CEN/TS 1992-4:2009 Part 2 and b) Acc. to Figure 10 of CEN/TS 1992-4:2009 Part 2

3.8.5 Selection of baseplate material dialog – baseplate design

The *Selection of baseplate material* dialog will be directly opened before the calculation/design of the baseplate. The selectable materials depend on the stud material. Figure 3.21 shows this dialog

1. **Radio Group *Material*:** The Radio Group *Material* allows to select the baseplate material depending on the stud material.
2. **Edit Box f_{yk} :** The read-only edit box f_{yk} shows the characteristic yield strength of the selected baseplate material.
3. **Edit Box $\gamma_{M,S}$:** The read-only edit box $\gamma_{M,S}$ shows the partial material safety factor used in the baseplate design.
4. **Button *Ok*:** If this button is clicked the user input data from the dialog will be used in the program and the dialog will be closed. After that the baseplate design (FE-Analysis) is started directly.

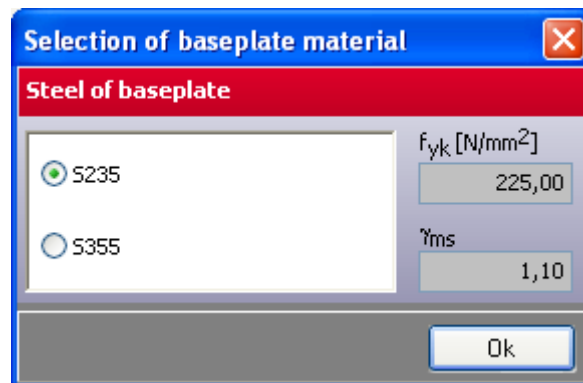


Figure 3.21: Selection of baseplate material dialog

3.8.6 Scope check information dialog

The Scope check information dialog (see Figure 3.22) shows informations about which scope check (boundary condition) is not fulfilled and why.

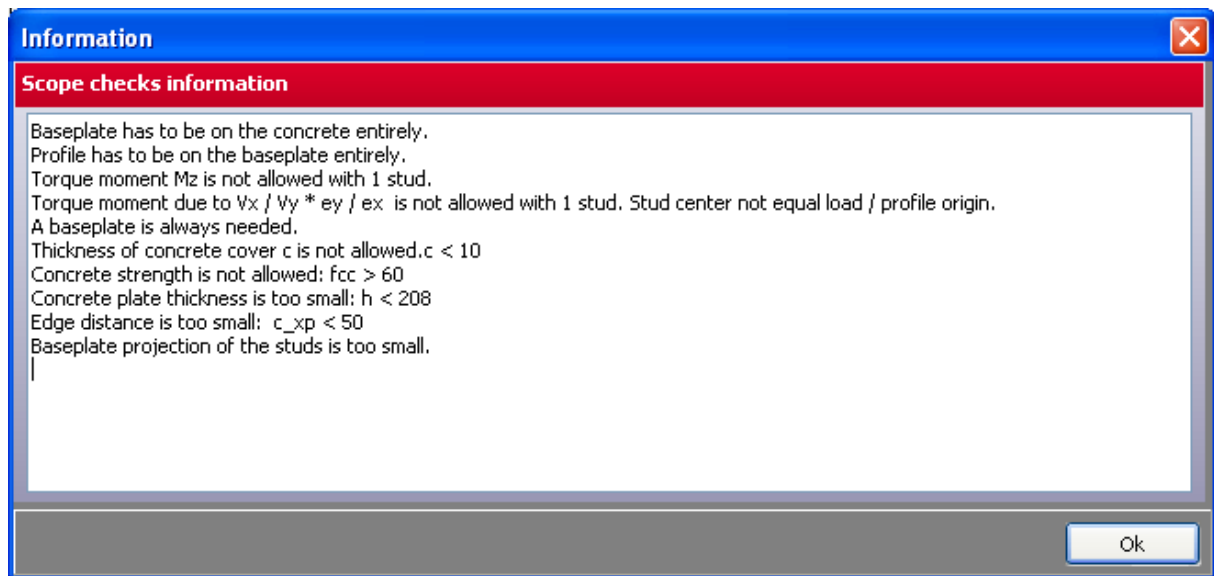


Figure 3.22: The Scope check information dialog

3.9 System dialogs

In the following chapter the functions of the system standard-dialogs that are used in the KÖCO StudCalc – Program are described. The language of the dialogs depends on the regional settings of the operating system (OS). More details about the *System dialogs* are given in the manual of the OS.

3.9.1 Open Dialog

The appearance of the *Open Dialog* depends on the operating system (OS) in use. See Figure 3.23 for an example. A project file (with extension ".hsd") can be chosen from the different media devices of the computer.

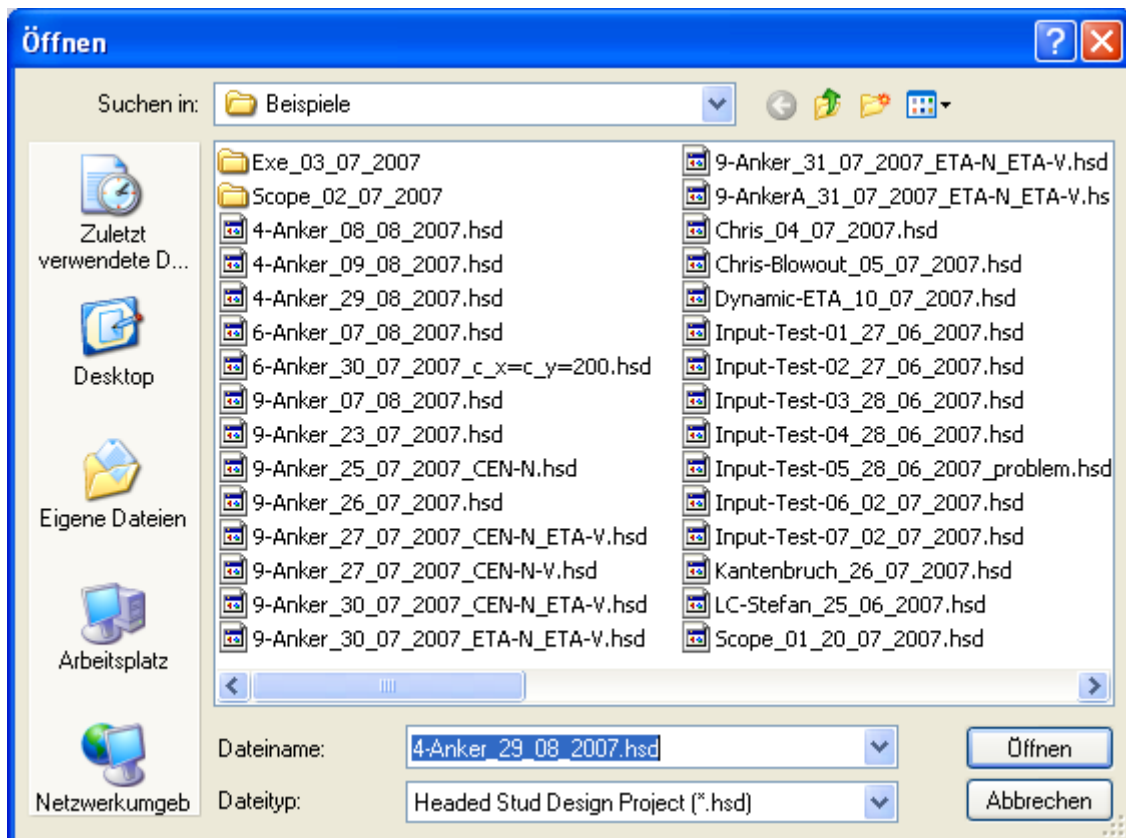


Figure 3.23: The Open Dialog

1. **Open:** The previously selected file will be opened.
2. **Cancel:** The dialog will be closed and no further actions will be performed.

3.9.2 Save Dialog

The appearance of the *Save Dialog* depends on the operating system in use. See Figure 3.24 for an example. A media device of the computer can be chosen for saving and a valid filename (*.hsd; the standard of the operating system must be maintained!) has to be defined.

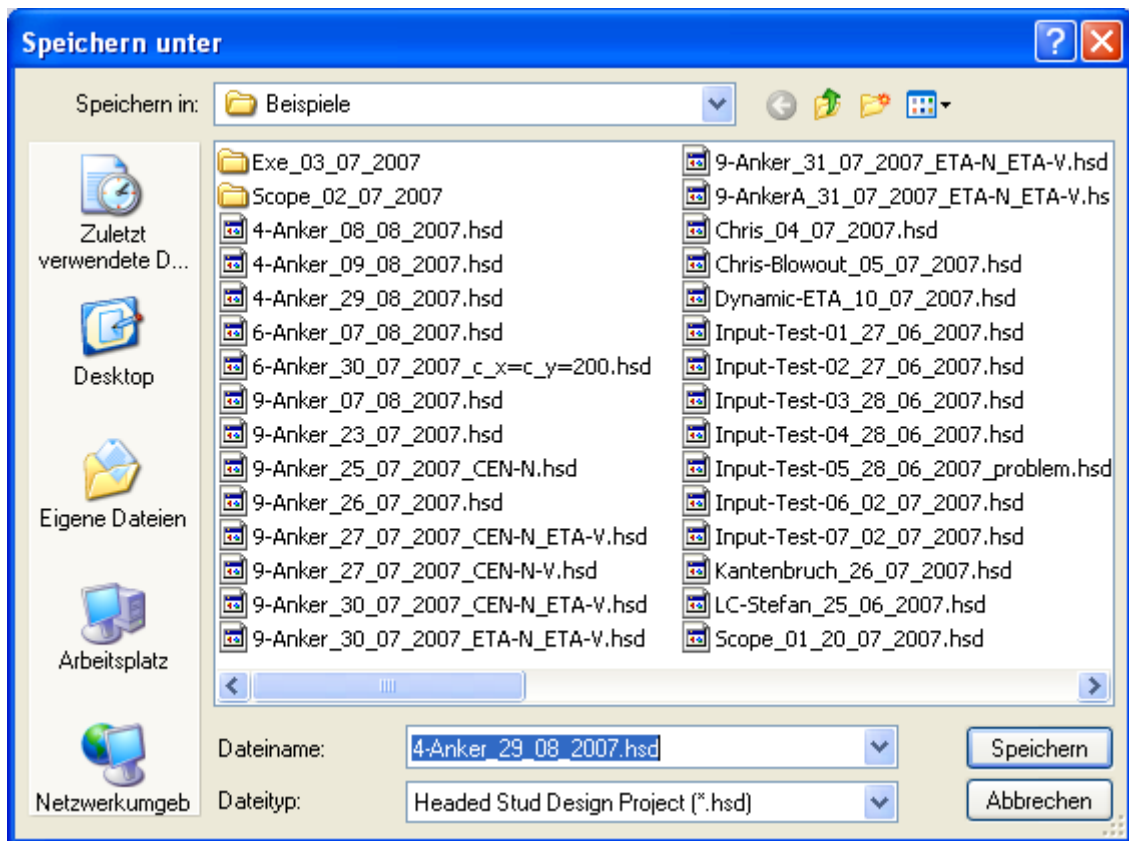


Figure 3.24: The Save Dialog

1. **Save:** The project (with extension ".hsd") will be saved on the previously selected media device with the defined filename.
2. **Cancel:** The dialog will be closed and no further actions will be performed.

3.9.3 Print Dialog

The appearance of the *Print Dialog* depends on the operating system in use. See Figure 3.25 for an example. An available printer on the system can be chosen. Several settings can be done. Note: **For the functionality of the print-preview a printer (could be a dummy!) or a printer driver has to be installed.**

1. **Ok:** The project report from the print-preview will be printed on the previously selected printer.
2. **Cancel:** The dialog will be closed and no further actions will be performed.

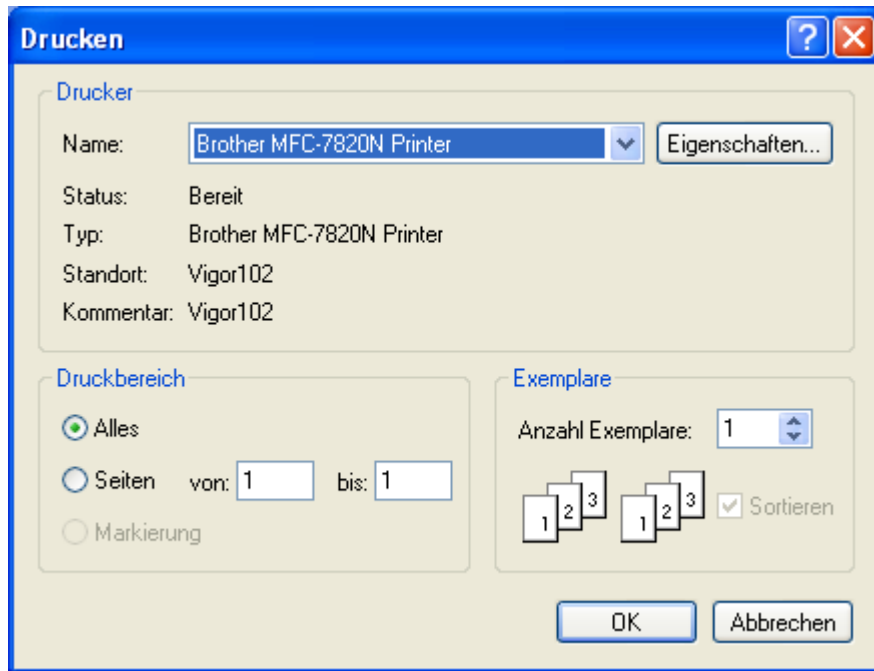


Figure 3.25: The Print Dialog

3.10 Database and stud selector

In this section the databases and the corresponding selector are described. There are two types of databases. A global *Products.mdb* that includes all data of the studs (KÖCO products) and profiles and a *Locale.mdb* for the translations into different languages.

3.10.1 Stud selector

The *Stud selector* is a dialog in which studs (under different selection criterias) can be selected from the database (see Figure 3.26). The dialog includes four areas. The *Selection criteria* area, where via a different ckeckboxes a user-defined preselection of the studs is possible, the *Available studs* list for selecting the studs, a *Button band* and the *Selected studs* list where the selected studs are shown.

Note: After the program start no check boxes are checked, this means that the list of available bolts is empty.

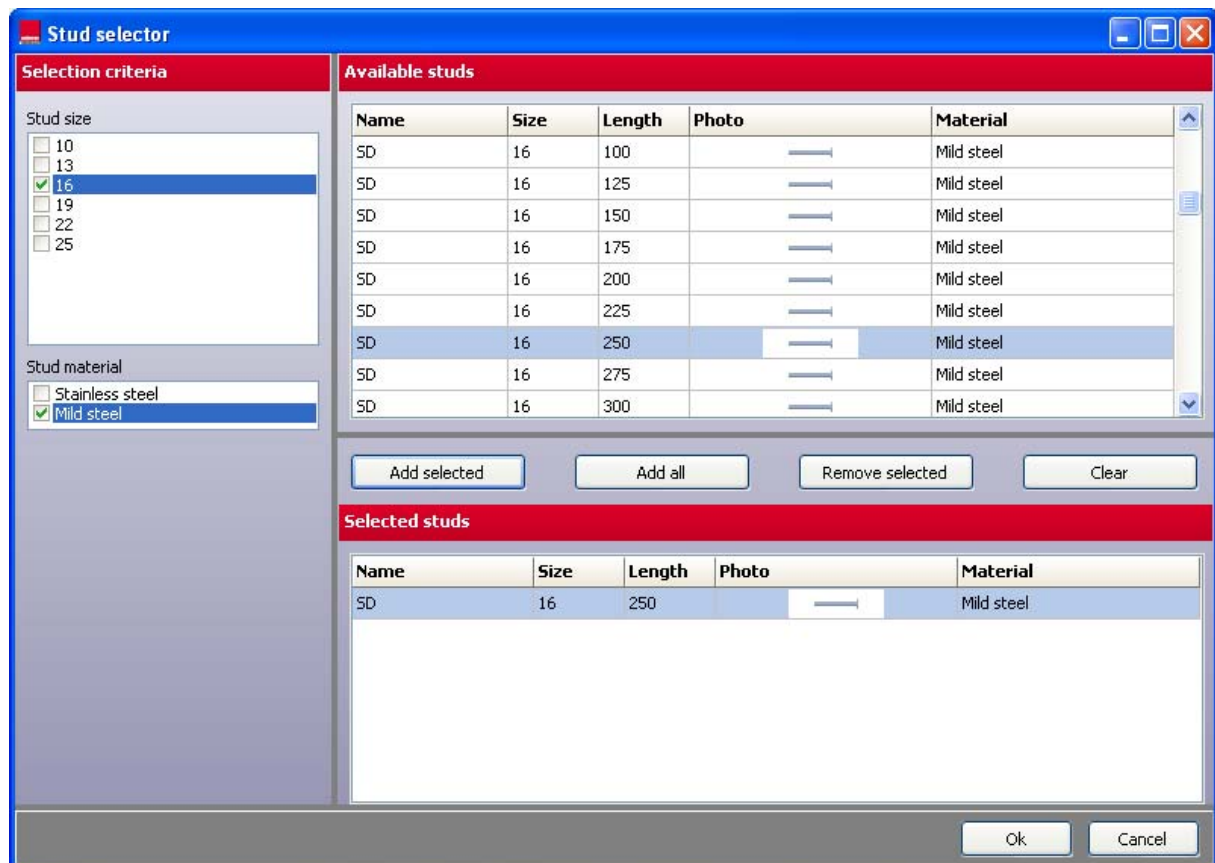


Figure 3.26: The Stud selector dialog

Selection criteria

In this area the criterias for the stud selection can be defined. *Stud size*: checkboxes for the size of the studs from 10 to 25. *Stud material*: checkboxes for stainless and mild steel

Available studs list / Selected studs list

The *Available studs* list and the *Selected studs* list, respectively, show a read-only table with columns representing the characteristic database entries (name, size, length) and a photo of the stud.

Based on the selections made in the *Selection criteria* area a list of studs which fulfill the criterias will be displayed in the *Available studs* list.

Button band

1. **Button Add selected:** If this button is clicked the currently selected stud (different colour) in the table of the *Available stud* list is chosen and displayed in the *Selected studs* list.
2. **Button Add all:** If this button is clicked all studs in the table of the *Available stud* list are chosen and displayed in the *Selected studs* list.

3. **Button Remove selected:** If this button is clicked the currently selected stud (different colour) in the table of the *Selected studs* list will be removed from the *Selected studs* list.
4. **Button Clear:** If this button is clicked all studs in the table of the *Selected studs* list will be removed.

General

1. **Button Ok:** If this button is clicked the currently selected studs in the table *Selected studs* list are chosen and the *Stud selector* is closed.
2. **Button Cancel:** If this button is clicked no studs are chosen and the *Stud selector* is closed.

3.11 Reports

There are three different kinds of reports that will be generated based on the calculation results: *normal force / proofs only*, *shear force / proofs only*, *normal and shear force / proofs (total)*. These kinds of reports are selected by the program internally. This means that only the necessary results will be shown. Results/Proofs which are not necessary are marked with “na” (= not applicable).

A sample report (*total*) is shown at the end of this section.

3.11.1 Report Preview

The *Report Preview* gives a preview of the print-out. See Figure 3.27 for details.

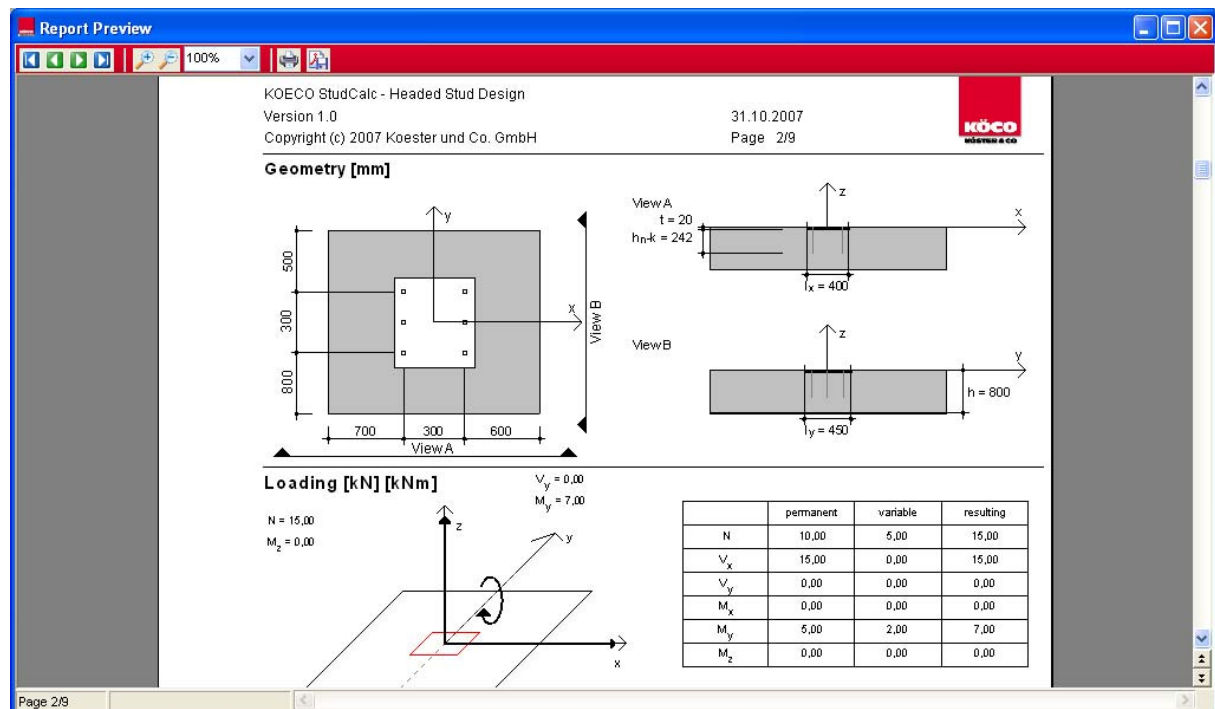


Figure 3.27: The Report Preview

The dialog consists of two areas. The upper area displays a set of buttons for controlling the *Preview Area*. The lower area shows the *Preview Area* containing the report. The preview is not fully WYSIWYG.

Control Buttons

1. **Button *First Page***: On click the first page of the report is shown.
2. **Button *Backward***: On click the page before the current page of the report is shown.
3. **Button *Forward***: On click the page after the current page of the report is shown.
4. **Button *Last Page***: On click the last page of the report is shown.
5. **Button *Zoom in***: On click the *Preview Area* will be zoomed in.
6. **Button *Zoom out***: On click the *Preview Area* will be zoomed out.
7. **Combo Box *Zoom***: Selection of a certain zoom percentage for the *Preview Area*.
8. **Button *Print***: The *Print Dialog* will be opened and the report can be printed. See chapter 3.9.3 for details about the *Print Dialog*.
9. **Button *Save to pdf***: The *SaveToPDF Dialog* will be opened and the report can be saved to a pdf-File. See chapter 3.9.2 for details about the *Save Dialog*.

Preview Area

The *Preview Area* shows a preview of the report. The *Report Preview* is not exactly WYSIWYG.

PDF-Report

An example of the report (total) follows.

Company

Name
Contact person
Address
Phone
Fax
email

Project-No.	12345
Project	Test-Projekt
Part	Part XYZ
Loadcase	

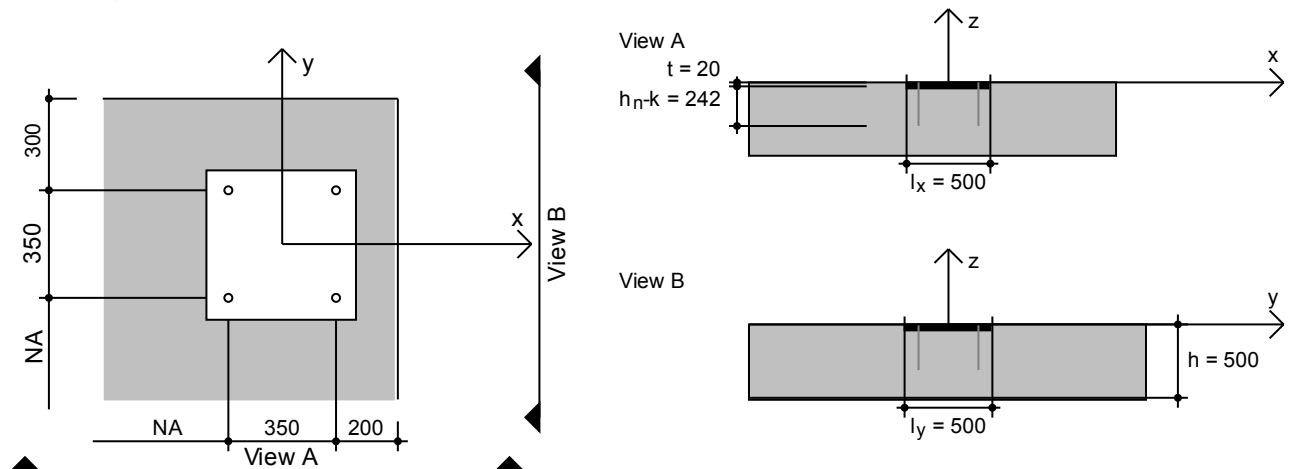
Notes

Test notes

Studname / material	SD-16x250, mild steel, S235J2+C450
Effective anchoring depth	$h_{ef} = 262$ [mm]; without consideration of base plate and member geometry
Approval No.	ETA-03/0039
Issued / Valid	31.05.2010/18.11.2013
Proof	Design method CEN/TS 1992-4:2009 Part 1 and 2 and ETA
Anchor plate	Rigid plate (calculation of stud forces); $l_x \times l_y \times t = 500 \times 500 \times 20$ mm
Concrete	cracked concrete C20/25, $f_{cc} = 25$ N/mm ² , $h = 500$ mm
Reinforcement	$d_s = 10$ mm, $s = 150$ mm, $c = 30$ mm, $f_{yk} = 500$ N/mm ² , crosswise: no, Close hangers ($a \leq 100$ mm): no
Supplementary reinforcement	Normal forces (Tension): no, Shear forces: no

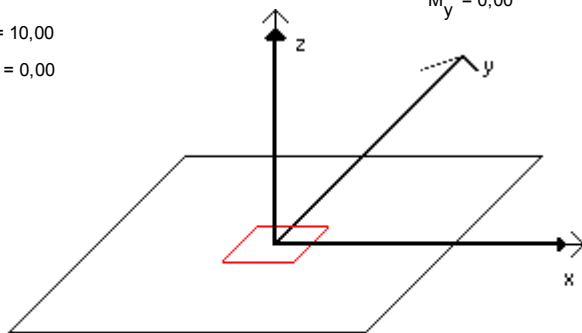


Geometry [mm]



Loading [kN] [kNm]

$N = 10,00$
 $M_z = 0,00$



$V_y = 0,00$
 $M_y = 0,00$

$V_x = 10,00$
 $M_x = 0,00$

Design loads	
N	10,00
V_x	10,00
V_y	0,00
M_x	0,00
M_y	0,00
M_z	0,00

Dynamic load percentage: 0,0 [%]

Eccentric profile [mm]:

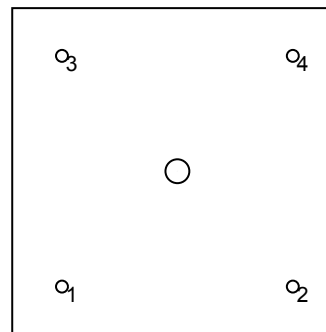
$e_x = 0$; $e_y = 0$

Design loads - F_{Rd}

Stud reactions [kN]

Tension force: (+ Tension; - Pressure)

Stud	Tension force	Shear force
1	2,50	2,50
2	2,50	2,50
3	2,50	2,50
4	2,50	2,50



Compression force

Tension force

Max. concrete compressive strain [%]: 0,00

Max. concrete compressive stress [N/mm²]: 0,00

Resulting tension force [kN]: 10,00

Resulting compression force [kN]: 0,00

Input data and results must be checked for agreement with the existing conditions, the national standards and for plausibility!

Tension load (CEN/TS 1992-4:2009 Part 2, Section 6.2 and ETA)

Proof	Design values [kN]		Utilisation β_N [%]	Status
	Load	Capacity		
Steel failure	2,50	67,14	4	OK
Pull-out failure	2,50	60,00	4	OK
Local blow-out failure, direction	na	na	na	na
Concrete cone failure	10,00	163,11	6	OK
Splitting failure	na	na	na	na

Steel failure

$N_{Rk,s}$ [kN]	$\gamma_{M,s}$	$N_{Rd,s}^h$ [kN]	N_{Sd}^h [kN]
94,00	1,400	67,14	2,50

Pull-out failure

$N_{Rk,p}$ [kN]	ψ_c	$\psi_{ucr,N}$	$\gamma_{M,p}$	$N_{Rd,p}^h$ [kN]	N_{Sd}^h [kN]
90,00	1,000	1,000	1,500	60,00	2,50

Local blow-out failure, direction

$A_{c,Nb}$ [mm ²]	$A_{c,Nb}^0$ [mm ²]	c [mm]	h_{ef}^* [mm]	n	s_1 [mm]
na	na	na	na	na	na
$\psi_{s,Nb}$	$\psi_{g,Nb}$	$\psi_{ec1,Nb}$	$\psi_{ec2,Nb}$	$\psi_{ucr,N}$	
na	na	na	na	na	
$N_{Rk,cb}^0$ [kN]		$\gamma_{M,c}$	$N_{Rd,cb}$ [kN]		N_{Sd} [kN]
na		na	na		na

Concrete cone failure

$A_{c,N}$ [mm ²]	$A_{c,N}^0$ [mm ²]	$c_{cr,N}$ [mm]	$s_{cr,N}$ [mm]	h_{ef}^* [mm]
983549	617796	393,00	786,00	262,00
$\psi_{s,N}$	$\psi_{re,N}$	$\psi_{ec1,N}$	$\psi_{ec2,N}$	
0,853	1,000	1,000	1,000	
$N_{Rk,c}^0$ [kN]	$\gamma_{M,c}$	$N_{Rd,c}$ [kN]	N_{Sd} [kN]	
180,24	1,500	163,11	10,00	

Note: not applicable proofs/values are marked with "na"!

*: Effective anchoring depth with consideration of base plate and member geometry!

Splitting failure (ETA, Annex 6)

A minimum reinforcement with the following cross-section shall exist in order to prevent a splitting of the concrete member:

$$A_{s,erf} = 0,5 * N_{sd} / (f_{yk} / \gamma_{Ms,re}) = 11 \text{ mm}^2$$

For slab- and plate-like members the reinforcement against splitting must be provided in both directions, i.e. in the case of members subjected mainly to tension the reinforcement shall be provided on both cross-sectional surfaces and in the case of members subjected mainly to bending on the side exposed to tension. It shall consist of at least three bars with a bar distance $\leq 150 \text{ mm}$ and shall be anchored, outside the anchorage, with an anchorage length according to national standards. For linear structures the splitting reinforcement needs to be provided in one direction only. In the case of anchorages near the edge of members this reinforcement must be also provided as edge reinforcement with corresponding hanger reinforcement.

Shear load (CEN/TS 1992-4:2009 Part 2, Section 6.3 and ETA)

Proof	Design values[kN]		Utilisation β_V [%]	Status
	Load	Capacity		
Steel failure	2,50	38,00	7	OK
Concrete pry out failure	10,00	326,22	3	OK
Concrete edge failure, direction X+	10,00	39,96	25	OK

Steel failure

$V_{Rk,s}$ [kN]	$\gamma_{M,s}$	$V_{Rd,s}^h$ [kN]	V_{Sd}^h [kN]
57,00	1,500	38,00	2,50

Concrete pryout failure (Bolt group relevant)

$A_{c,N}$ [mm ²]	$A_{c,N}^0$ [mm ²]	$c_{cr,N}$ [mm]	$s_{cr,N}$ [mm]	h_{ef}^* [mm]	k_3
983549	617796	393,00	786,00	262,00	2,0
$\psi_{s,N}$	$\psi_{re,N}$	$\psi_{ec1,V}$	$\psi_{ec2,V}$		
0,853	1,000	1,000	1,000		
$N_{Rk,c}^0$ [kN]	$\gamma_{M,c,p}$	$V_{Rd,cp}$ [kN]	V_{Sd} [kN]		
180,24	1,500	326,22	10,00		

Concrete edge failure, direction X+

l _f [*] [mm]		d ₁ [mm]	c ₁ [mm]	A _{c,V} [mm ²]	A _{c,V} ⁰ [mm ²]
128,00		16,00	200,00	285000	180000
ψ _{s,V}		ψ _{h,V}	ψ _{ec,V}	ψ _{α,V}	ψ _{re,V}
1,000		1,000	1,000	1,000	1,000
α	β	V _{Rk,c} ⁰ [kN]	γ _{M,c}	V _{Rd,c} [kN]	V _{Sd} [kN]
0,080	0,060	37,85	1,500	39,96	10,00

Note: not applicable proofs/values are marked with "na"!

*: Effective anchoring depth with consideration of base plate and member geometry!

Combined tension and shear loads (CEN/TS 1992-4:2009 Part 2, Section 6.4 and ETA)

β_N	β_V	α	Utilisation $\beta_{N,V}$ [%]	Status
0,061	0,250	1,5	14	OK

$$\beta_N^\alpha + \beta_V^\alpha \leq 1$$

Range of steel stresses (not predominantly static loads, ETA section 4.2)

Proof	Present stress range [N/mm ²]	Allowed stress range [N/mm ²]	Utilisation [%]	Status
Tension	$\Delta \sigma_N = \text{na}$	$\Delta \sigma_{N,zul} = \text{na}$	na	na
Shear	$\Delta \sigma_V = \text{na}$	$\Delta \sigma_{V,zul} = \text{na}$	na	na
Supplem. reinforcement	$\Delta \sigma_{re} = \text{na}$	$\Delta \sigma_{re,zul} = \text{na}$	na	na

Proof of the transmission of the anchor loads to the supports

Transmission of the anchor loads into the concrete

Checking the transfer of loads into the base is required in accordance with CEN/TS 1992-4:2009 Part 1, Annex A!

Shear resistance of base material

Shear resistance of the base material must be checked according to CEN/TS 1992-4:2009 Part 1, Annex A and EN 1992-1-1 (or National Standards).

Fastening OK!

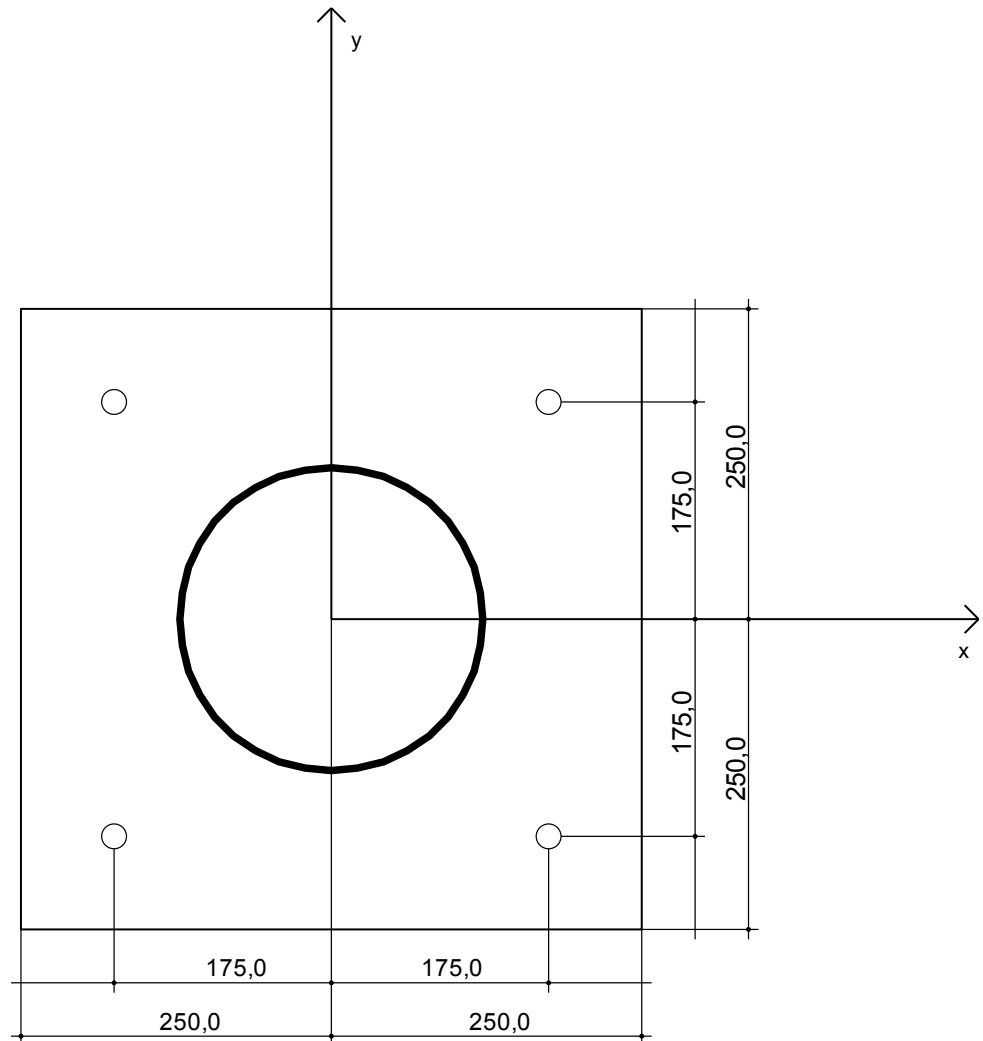
Note: not applicable proofs/values are marked with "na"!

Anchor plate, steeltype: S235

Profile type: Tube - 244,5 x 6,3 - (245 x 245 x 6)

Input plate thickness: 20 [mm]

Calculated plate thickness: 8 [mm]



Material list

Notation	Measures [mm]	No. of	Weight [kg]
Steel plate	500 x 500 x 20	1	39,25
Studs (016-0153)	16 x 250	4	1,75

Input data and results must be checked for agreement with the existing conditions, the national standards and for plausibility!