PROFIS Engineering Suite

English

PROFIS Engineering Suite

Original operating instructions

***** For internal use only ***** 13.12.2018 / 14:10:21 - H2\DOK-Projekt\System\PROFIS_Engineering\ENU\PROFIS_ING

Contents

***** For internal use only ***** 13.12.2018 / 14:10:21 - H2\DOK-Projekt\System\PROFIS_Engineering\ENU\PROFIS_ING

***** For internal use only ***** 13.12.2018 / 14:10:21 - H2\DOK-Projekt\System\PROFIS_Engineering\ENU\PROFIS_ING

1 Information about the documentation

1.1 Symbols in the documentation

1.1.1 Explanation of symbols

The following symbols are used on the product:

	Home
	Settings
	Report template
	Info

***** For internal use only ***** 13.12.2018 / 14:10:21 - H2\DOK-Projekt\System\PROFIS_Engineering\ENU\PROFIS_ING

⋮	Pop-up menu
=	Show / hide
⌄	Show / hide

1.2 Product information

Hilti software is designed for professional use and may be used and maintained only by trained, authorized personnel. This personnel must be informed of any particular hazards that may be encountered.

- Read this documentation before initial operation or use. This is a prerequisite for safe, trouble-free handling and use of the product.
- Observe the safety instructions and warnings in this documentation and on the product.
- You will be required to state the product details when contacting Hilti Service or your local Hilti organization to enquire about the product.
  Product information → page 4

Product information

Software	PROFIS Engineering Suite
Version	3.0.16

1.3 System requirements

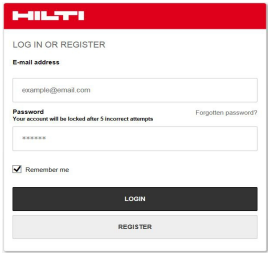
Information about system requirements can be found at:
qr.hilti.com/r6502279.

2 Description

Read this documentation before initial operation or use.
PROFIS Engineering Suite is software designed to find the correct anchor for an application. In addition, PROFIS Engineering Suite can be used to calculate the matching anchor, base plate and railing combination. PROFIS Engineering Suite is available in online and offline versions.

3 Log in

1. To log in as a user, enter the following web address in your browser: https://profisengineering.hilti.com/.
2. Enter your login data:
   - ➥ E-mail address
   - ➥ Password
3. Click **'Login'**.
   - ➥ The **'PROFIS Engineering Suite'** window opens.

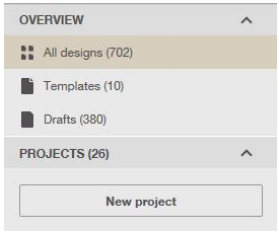


***** For internal use only **** 13.12.2018 / 14:10:21 - H2\DOK-Projekt\System\PROFIS_Engineering\ENU\PROFIS_ING

4 Managing projects

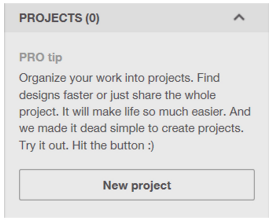
4.1 Creating a new project

1. Click one of the following project views:
   - ➥ **'All designs'**
   - ➥ **'Templates'**
   - ➥ **'Drafts'**
2. Click the **'New project'** button.
   - ➥ An input box is displayed.
3. Type the name of the project in the input box.
4. Confirm the name of the project by pressing this button ✓ or cancel the operation by pressing this button ×.
   - ➥ Your project appears in the project list.



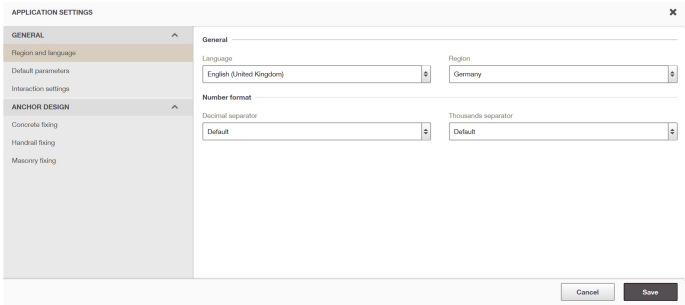
4.2 Editing a project

1. Highlight the project of your choice.
   - ➥ The Project menu appears ⋮.
2. Click on the Project menu and select one of the following options:
   - ➥ **'Add sub project'** → page 5
   - ➥ **'Rename'**
   - ➥ **'Archive'**



5 'Settings'

5.1 'Settings'



1. Select the appropriate values from the drop-down menu in the **'General'** box.
2. Select the appropriate values from the drop-down menu in the **'Number format'** box.
3. Click the **'Save'** button to confirm the entries.
4. Click the **'Cancel'** button to reject the entries.

5.2 'Default settings'

1. Type the corresponding parameters in the input boxes in the **'Default settings'** section.
2. Click the **'Save'** button to confirm the entries.
3. Click the **'Cancel'** button to reject the entries.

***** For internal use only ***** 13.12.2018 / 14:10:21 - H2\DOK-Projekt\System\PROFIS_Engineering\ENU\PROFIS_ING

5.3 'Quick-start configuration'

5.3.1 'Concrete fixing'

1. Select the corresponding value from the drop-down menu in the **'General'** section.
2. Select the corresponding values from the drop-down menu in the **'Units and default parameters'** section.
3. Activate the corresponding option buttons in the **'Calculation method and approvals'** section.

5.3.2 'Masonry fixing'

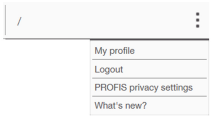
1. Select the corresponding value from the drop-down menu in the **'General'** section.
2. Select the corresponding values from the drop-down menu in the **'Number format'** section.
3. Activate the corresponding option buttons in the **'Calculation method and approvals'** section.

5.3.3 'Handrail fixing'

1. Select the corresponding value from the drop-down menu in the **'General'** section.
2. Select the corresponding values from the drop-down menu in the **'Number format'** section.
3. Activate the corresponding option buttons in the **'Calculation method and approvals'** section.

6 The 'My profile' menu

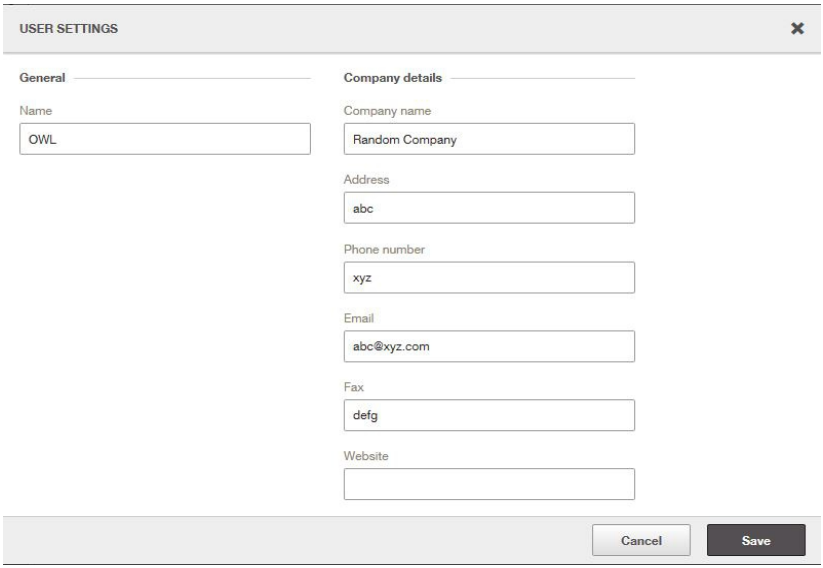
- Click on the **'My profile'** menu.
  ➥ A menu selection appears.



6.1 Logging out

- Click on **'Logout'**.
  ➥ You have logged out.

6.2 Editing the profile



1. Click and select **'My profile'**.
   ➥ The **'User settings'** window appears.
2. Complete the input boxes.

***** For internal use only ***** 13.12.2018 / 14:10:21 - H2\DOK-Projekt\System\PROFIS_Engineering\ENU\PROFIS_ING

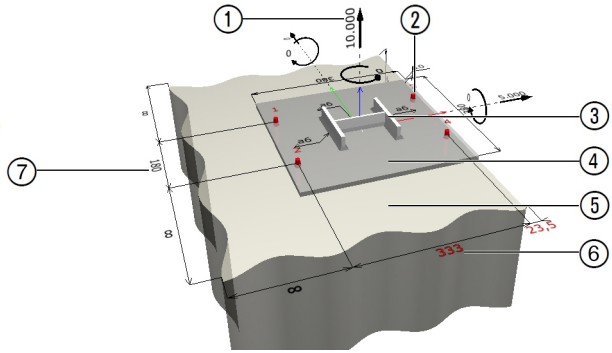
3. Click **‘Save’** to confirm the entries.
4. To cancel the entry you have made, click **‘Cancel’**.

7 Editor

7.1 Editor menu

	Undo
	Redo
	Reset camera
	Display

7.2 3D editor



① Loads
② Anchor
③ Profiles
④ Anchor plate
⑤ Base material
⑥ Unit of measure (outside the recommended range.)
⑦ Unit of measure

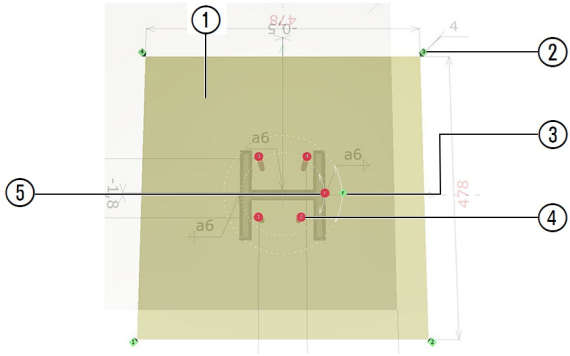
7.2.1 3D editor

1. To define a value (unit of measure or force), click on the corresponding value.
   ➥ An input box appears
2. Enter the appropriate value.
3. Press the Enter key.

A red number indicates that the value is too high or too low.

4. To show or hide dimensions, click on the 👁 button.

***** For internal use only ***** 13.12.2018 / 14:10:21 - H2\DOK-Projekt\System\PROFIS_Engineering\ENU\PROFIS_ING



① Anchor plate
② Positioning the edge of the anchor plate
③ Rotating the anchor plate
④ Positioning the anchor
⑤ Rotating the anchor

7.3.1 'Coordinate center'

- Enter the coordinates in the corresponding tables **'Anchor plate nodes'** and **'Anchor nodes'**.
  ➥ The new position will be shown in the 2D editor.

To shift the position markers, click one of the markers and drag it to the desired position.

7.3.2 'Custom layout'

1. Click **'Create'** to confirm the entries.
2. Click **'Cancel'** to reject the entries.

7.3.3 2D editor

1. To define a value (unit of measure or force), click on the corresponding value.
   ➥ An input box appears
2. Enter the appropriate value.
3. Press the Enter key.

A red number indicates that the value is too high or too low.

4. To show or hide dimensions, click on the 👁 button.
5. To do directly to the desired settings, double-click on the corresponding element (e.g. anchor, anchor plate, base material).

8 Quick start

8.1 Fastening on concrete

8.1.1 Creating a favorites list

8.1.1.1 'My favorite inputs'

- To add a menu to the favorites inputs list, click this symbol ☆ for the desired menu.
  ➥ The selected menu appears in the list of favorites.

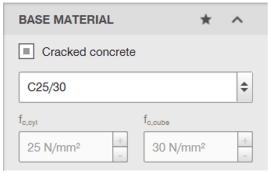
To remove a menu from the favorite inputs list, click this option ★.

***** For internal use only ***** 13.12.2018 / 14:10:21 - H2\DOK-Projekt\System\PROFIS_Engineering\ENU\PROFIS_ING

8.1.2 Defining the base material

8.1.2.1 'Base material'

1. If the base material is cracked, activate the **'Cracked concrete'** option button.
2. Select the applicable concrete grade from the drop-down menu.



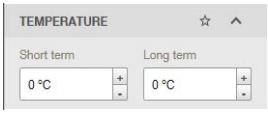
8.1.2.2 'Temperature'

1. In the **'Short term'** input box, enter the temperature to which the base material can be exposed for a short time.

   A short-term influence is understood to mean, for example, the temperature difference between day and night.

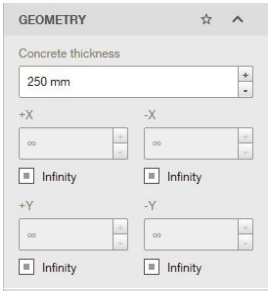
2. In the **'Long term'** input box, enter the long-term temperature influences.

   Long-term temperature influences are the base material's lasting temperature characteristics.



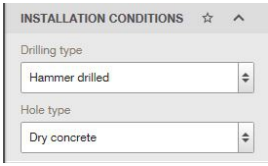
8.1.2.3 'Geometry'

1. Enter the thickness of the component in the **'Concrete thickness'** input box.
2. Enter the edge distance in relation to directions **'+X'**, **'-X'**, **'+Y'** and **'-Y'** in the corresponding input boxes.
3. To set the edge distance to "infinite", activate the **'Infinite'** checkbox.



8.1.2.4 'Installation conditions'

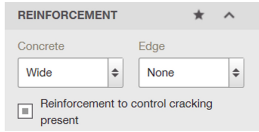
1. Select the drilling method from the **'Drilling method'** drop-down menu.
2. Select the condition of the drilled hole from the **'Hole type'** drop-down menu.



***** For internal use only ***** 13.12.2018 / 14:10:21 - H2\DOK-Projekt\System\PROFIS_Engineering\ENU\PROFIS_ING

8.1.2.5 'Reinforcement'

1. Select the spacing between reinforcing bars from the **'Concrete'** drop-down menu.



> The spacing is understood to be "wide" when the distance between bars is ≥150 mm, with any diameter, or when the distance is ≥100 mm with a diameter of ≤10 mm.

2. Select the spacing between rebars from the **'Edge'** drop-down menu.

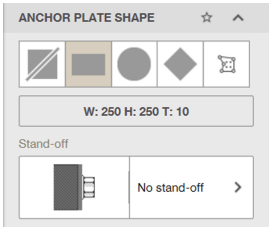
> Straight edge reinforcement assumes edges reinforced with a rebar diameter of ≥12 mm. The hairpin reinforcement option assumes edge reinforcement with a bar diameter of ≥12 mm and closely spaced hairpin rebars at intervals of ≤100 mm. The edge reinforcement is taken into account in the verification of concrete edge breakage as a result of concrete cracking due to shear loading.

3. Activate the **'Reinforcement to control cracking present'** option button if the reinforced concrete is cracked. The maximum width of the cracks is approximately 3 mm.

8.1.3 Defining the anchor plate

8.1.3.1 'Anchor plate shape'

1. Click the shape of the anchor you wish to use.
   ➥ The corresponding shape appears in the 3D editor.

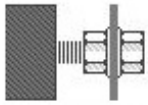


> To define the shape yourself, click the button.
> Use the 2D editor to edit the shape of the anchor plate.

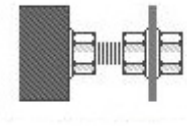
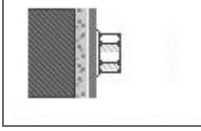
2. To enter the dimensions of the anchor plate, click the W: 140 H: 140 T: 10 button.
   ➥ The **'Anchor plate size'** window is displayed.
3. Type the required parameters in the corresponding input boxes.
4. Click **'Save'** to confirm the entries.
5. Click **'Cancel'** to cancel the operation.

8.1.3.2 'Stand-off type'

- Select the type of stand-off installation.

	**'Not stand-off'**
	**'Stand-off without clamping'**

***** For internal use only **** 13.12.2018 / 14:10:21 - H2\DOK-Projekt\System\PROFIS_Engineering\ENU\PROFIS_ING

	**‘Stand-off with clamping’**
	**‘Stand-off with grounding’**

8.1.3.3 ‘Anchor plate thickness’

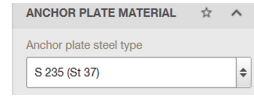
- Enter the thickness in the **‘Thickness’** box and press Enter to confirm your entry.

8.1.3.4 ‘Anchor plate design’

1. To show the load, activate the **‘Show normal stress distribution on 3D’** option button.
2. To optimize the anchor plate thickness, activate the **‘Show optimized anchor plate thickness’** option button.

8.1.3.5 ‘Anchor plate material’

- Select the steel grade you are using from the drop-down box.



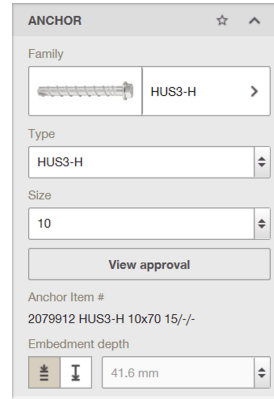
8.1.4 Defining the anchor

8.1.4.1 ‘Anchor’

1. To select the anchor type, click on the **‘Family’** box.
   ➥ The **‘Select anchor’** window appears.

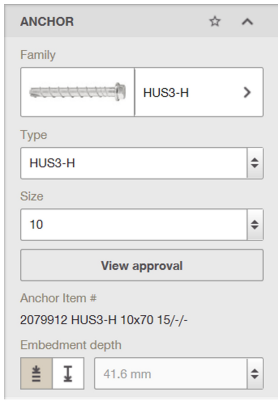
   Alternatively, you can click on the **‘Family’** box in the main window to select the anchor type. If you want an anchor to be first in the **‘Select anchor’** window you can highlight it as a favorite.
2. To search for an anchor, enter its name in the search box.
3. To sort the list of anchors, select the sorting criterion of your choice from the drop-down menu.
4. To calculate all anchors, click on the **‘Calculate all’** button.
   ➥ The degree of utilization and the geometry of the anchors are calculated and shown in the list.

   Click on **‘Clear’** to return to the normal view.
5. To define the list of anchors, activate the appropriate radio button in the **‘Filter’** box.



***** For internal use only ***** 13.12.2018 / 14:10:21 - H2\DOK-Projekt\System\PROFIS_Engineering\ENU\PROFIS_ING

6. To filter the list of anchors by **'Fixture thickness'** or **'Hole diameter'**, enter the values in the **'max.'** and **'min.'** input boxes.
7. To filter the list of anchors by one of the following criteria, activate the appropriate radio button:
   - ➥ **'Anchor type'**
   - ➥ **'Corrosion / material'**
   - ➥ **'Cleaning'**
   - ➥ **'Setting'**
   - ➥ **'Thread type'**
   - ➥ **'Head configuration'**
   - ➥ **'Installation type'**
8. Select the anchorage depth in the **'Embedment depth'** box.
   - ➥ Optimized embedment depth – .
   - ➥ User-selected embedment depth – .



Optimized anchorage depth – **Hilti** Anchor Installer determines the anchorage depth for maximum anchor loading capacity.

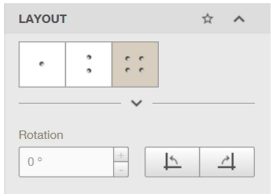
User-defined anchorage depth – the user can specify the required anchorage depth. For adhesive mortar (chemical) anchors, all values are possible. For metal anchors, discrete values must be adhered to. You can select the appropriate depth here from the drop-down menu.

8.1.4.2 'Layout'

1. Select the number and layout of the anchors in the **'Layout'** section.

To apply a user-defined layout, click on the button.

The 2D editor appears.



2. Enter the value by which you want to rotate the anchor plate In the **'Rotation'** input box.
3. To rotate the anchor plate 90° counter-clockwise, click on the button.
4. To rotate the anchor plate 90° clockwise, click on the button.

8.1.4.3 'Installation'

- ▸ To display the filled annular gap in the 3D editor, click the **'Filled holes'** button.

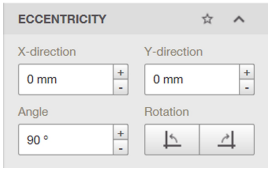
***** For internal use only ***** 13.12.2018 / 14:10:21 - H2\DOK-Projekt\System\PROFIS_Engineering\ENU\PROFIS_ING

8.1.5 Defining the steel profile

8.1.5.1 'Profile'

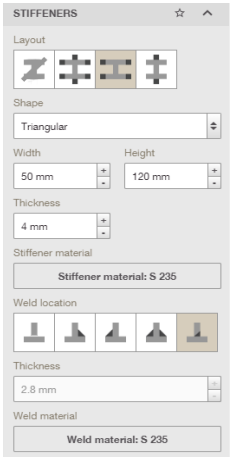
1. To define the rail profile, click on the **'Rail profiles'** button.
   - ➥ The **'Select profile'** window is displayed.
2. To select the dimensions, click on the dimensions of the rail profile.
   - ➥ The **'Select profile'** window is displayed.
3. To select the dimensions, click on the dimensions of the rail profile.
   - ➥ The **'Select profile'** window is displayed.
4. To define the post profile, click on the **'Post profiles'** button.
   - ➥ The **'Select profile'** window is displayed.
5. To select the dimensions, click on the dimensions of the post profile.
   - ➥ The **'Select profile'** window is displayed.
6. Click on the appropriate profile size.

8.1.5.2 'Eccentricity'



1. To shift the steel profile, (X-/ and Y-axis), type the corresponding values in the **'X-direction'** and **'Y-direction'** input boxes.
2. To rotate the steel profile, enter the corresponding value in the **'Angle'** box.
3. To rotate the anchor plate to the left through 90°, click the button.
4. To rotate the anchor plate to the right through 90°, click the button.

8.1.5.3 'Stiffeners'

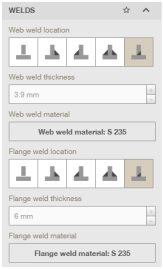


1. To select the material, click on the **'Materials'** button.
   - ➥ Select the appropriate material.
2. To select the shape of the stiffeners, click on the **'Shape'** button and select the appropriate shape:
   - ➥ **'Rectangular'**
   - ➥ **'Triangular'**
   - ➥ **'Chamfered'**
3. To select the thickness, click on the **'Thickness'** button and set the thickness in millimeters.
4. To select the layout of the stiffeners, click on the **'Layout'** button and select the layout of your choice.
5. Enter the dimensions **'Width'** and **'Height'** in the corresponding input boxes.
6. If you want to use welded stiffeners, select the appropriate option. If you use welded stiffeners, also select the appropriate entries:
   - ➥ **'Thickness'**
   - ➥ **'Materials'**
   - ➥ **'Layout'**

***** For internal use only ***** 13.12.2018 / 14:10:21 - H2\DOK-Projekt\System\PROFIS_Engineering\ENU\PROFIS_ING

8.1.5.4 Weld on webs and flanges

1. Select the appropriate option, either for welding on webs or for welding on flanges.
2. Select the thickness.
3. Select the material.
4. Select the weld location.



8.1.6 Defining the loads

8.1.6.1 Anchor plate design approach

1. Select **'Flexible'** for flexible effects.
2. Select **'Rigid'** for rigid effects.

8.1.6.2 'Load type'

1. For static actions, select **'Static or quasi-static design'**.

   The load takes the action of static and predominantly stationary live loads into account.

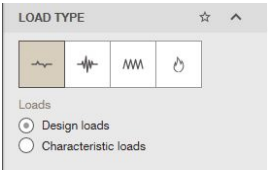
2. For seismic actions, select **'Seismic design'**.

   The load takes into account the current European EOTA TR 045 regulations for anchors designed to take up seismic actions.

3. For fatigue loads, select **'Fatigue design'**.

   Fatigue loads are loads that occur frequently during the life cycle of a building or structure.

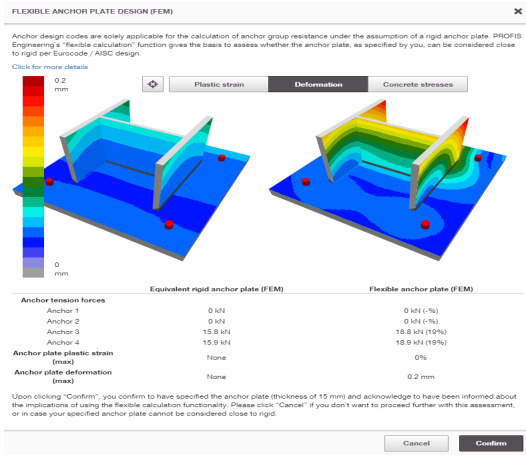
4. For influence of fire, select **'Fire'**.
5. Select one of the following option buttons:
   - **'Design loads'**
   - **'Characteristic loads'**



'Design loads' – Enter the loads calculated at the design level.
'Characteristic loads' – Enter the values for constant and variable loads. In PROFIS Engineering Suite these values are multiplied by the partial safety factors defined under **'Project options'**.

***** For internal use only **** 13.12.2018 / 14:10:21 - H2\DOK-Projekt\System\PROFIS_Engineering\ENU\PROFIS_ING

8.1.7 Calculate



1. Click on the "Calculate" button.
   - ➥ The calculations are displayed.
   - ➥ The colors indicate the severity of the loads.
2. Select the load of your choice. You can choose between:
   - ➥ Plastic strain
   - ➥ Deformation
   - ➥ Concrete stresses
3. Check the loads.
4. Close the dialog by selecting one of the two options:
   - ➥ Confirm
   - ➥ You accept the flexibility of the base plate and the results are displayed.
   - ➥ Cancel
   - ➥ You are taken back to the previous screen, where you can make changes that will lead to a higher degree of rigidity.
5. If the deviations are excessive a pop-up opens, notifying you that the deviations are large.

8.2 Fastening on masonry

8.2.1 Creating a favorites list

8.2.1.1 'My favorite inputs'

- ▸ To add a menu to the favorites inputs list, click this symbol ☆ for the desired menu.
  - ➥ The selected menu appears in the list of favorites.

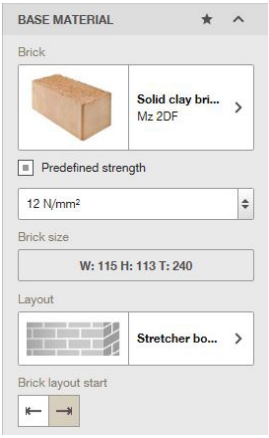
> To remove a menu from the favorite inputs list, click this option ★.

***** For internal use only ***** 13.12.2018 / 14:10:21 - H2\DOK-Projekt\System\PROFIS_Engineering\ENU\PROFIS_ING

8.2.2 Defining the base material

8.2.2.1 'Base material'

1. To select the type of brick or block, click the **'Brick'** button.
   ➥ The **'Select brick'** window is displayed.
2. To define the type of brick or block, click the applicable brick/block.
3. To define a predefined strength, activate the **'Predefined strength'** option button.



4. To define the dimensions of the brick or block, click the **'Brick size'** button.
   ➥ The **'Brick size'** window is displayed.
5. Enter the dimensions **'Width' 'Height'** and **'Concrete thickness'** in the corresponding input boxes.
6. Click the **'Save'** button to confirm the dimensions you have entered.
7. Click the **'Cancel'** button to reject the values you have entered.
8. Click the **'Layout'** button to define the layout of the bricks/blocks.
   ➥ The **'Select layout'** window is displayed.
9. Click the corresponding illustration to select the layout of the bricks/blocks.
10. Click one of the following buttons to specify the original position of the masonry bricks/blocks:
    ➥ 
    ➥ 

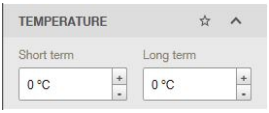
8.2.2.2 'Temperature'

1. In the **'Short term'** input box, enter the temperature to which the base material can be exposed for a short time.

   A short-term influence is understood to mean, for example, the temperature difference between day and night.

2. In the **'Long term'** input box, enter the long-term temperature influences.

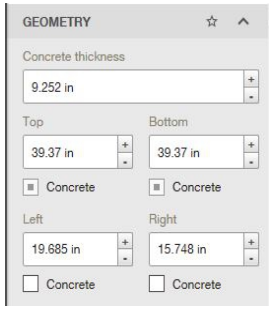
   Long-term temperature influences are the base material's lasting temperature characteristics.



***** For internal use only **** 13.12.2018 / 14:10:21 - H2\DOK-Projekt\System\PROFIS_Engineering\ENU\PROFIS_ING

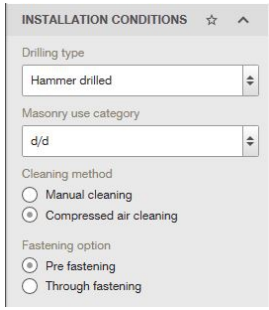
8.2.2.3 'Geometry'

1. Enter the edge distance with reference to the directions **'Top'**, **'Bottom'**, **'Left'** and **'Right'** on the drawing.
2. Activate the **'Concrete'** option button in order to display, in the 3D editor, the value you have entered for "concrete".



8.2.2.4 'Installation conditions'

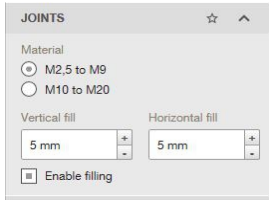
1. Select the drilling method from the **'Drilling method'** drop-down menu.
2. Select the utilization category with regard to installation and use in accordance with ETA from the **'Masonry use category'** drop-down menu:
   - d = dry
   - w= wet



3. To select the hole cleaning method, activate the corresponding option button in the **'Cleaning method'** section.
4. To select the fastening method, activate the corresponding option button in the **'Fastening option'** box.

8.2.2.5 'Joints'

1. In the **'Materials'** section, activate the corresponding option button for the joint material used.
2. Enter the width of the vertical joints in the **'Vertical fill'** input box.
3. Enter the width of the horizontal joints in the **'Horizontal fill'** input box.
4. Activate the **'Enable filling'** option button if the butt joints are mortar joints.



8.2.2.6 'Plaster'

- If the masonry is plastered/rendered, activate the **'Plastered wall'** option button.

***** For internal use only ***** 13.12.2018 / 14:10:21 - H2\DOK-Projekt\System\PROFIS_Engineering\ENU\PROFIS_ING

8.2.3 Defining the anchor plate

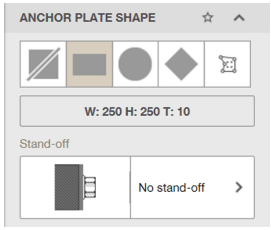
8.2.3.1 'Anchor plate position'

1. In the **'Rotation'** input box, enter the value by which you wish to rotate the anchor plate.
2. To rotate the anchor plate to the left through 90°, click the button.
3. To rotate the anchor plate to the right through 90°, click the button.

8.2.3.2 'Anchor plate shape'

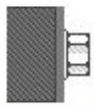
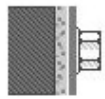
1. Click the shape of the anchor you wish to use.
   ➥ The corresponding shape appears in the 3D editor.

   To define the shape yourself, click the button.
   Use the 2D editor to edit the shape of the anchor plate.
2. To enter the dimensions of the anchor plate, click the W: 140 H: 140 T: 10 button.
   ➥ The **'Anchor plate size'** window is displayed.
3. Type the required parameters in the corresponding input boxes.
4. Click **'Save'** to confirm the entries.
5. Click **'Cancel'** to cancel the operation.



8.2.3.3 'Stand-off type'

- Select the type of stand-off installation.

	**'Not stand-off'**
	**'Stand-off with grounding'**

8.2.3.4 'Anchor plate thickness'

- Enter the thickness in the **'Thickness'** box and press Enter to confirm your entry.

***** For internal use only ***** 13.12.2018 / 14:10:21 - H2\DOK-Projekt\System\PROFIS_Engineering\ENU\PROFIS_ING

8.2.4 Defining the anchor

8.2.4.1 'Anchor'

1. To select the anchor type, click on the **'Family'** box.
   ➥ The **'Select anchor'** window appears.

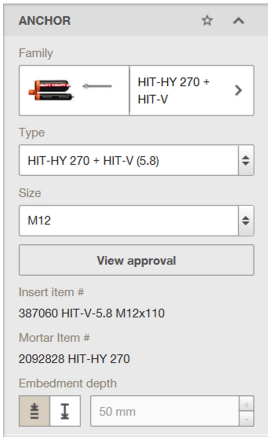
   Alternatively, you can click on the **'Family'** box in the main window to select the anchor type.
   If you want an anchor to be first in the **'Select anchor'** window you can highlight it as a favorite.

2. To search for an anchor, enter its name in the search box.
3. To sort the list of anchors, select the sorting criterion of your choice from the drop-down menu.
4. To calculate all anchors, click on the **'Calculate all'** button.
   ➥ The degree of utilization and the geometry of the anchors are calculated and shown in the list.

   Click on **'Clear'** to return to the normal view.

5. To filter the list of anchors by one of the following criteria, activate the appropriate radio button:
   ➥ **'Corrosion / material'**
   ➥ **'Thread type'**
6. To view the approval, click on the **'View approval'** button.
   ➥ The approval appears.
7. Select the anchorage depth in the **'Embedment depth'** box.
   ➥ Optimized depth – .

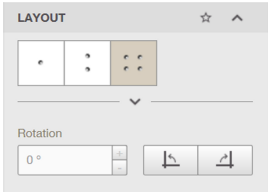
   Anchorage depth – the user can specify the required anchorage depth. For adhesive mortar (chemical) anchors, all values are possible. For metal anchors, discrete values must be adhered to. You can select the appropriate depth here from the drop-down menu.



8.2.4.2 'Layout'

1. Select the number and layout of the anchors in the **'Layout'** section.

   To apply a user-defined layout, click on the button.
   The 2D editor appears.



2. Enter the value by which you want to rotate the anchor plate In the **'Rotation'** input box.
3. To rotate the anchor plate 90° counter-clockwise, click on the button.
4. To rotate the anchor plate 90° clockwise, click on the button.

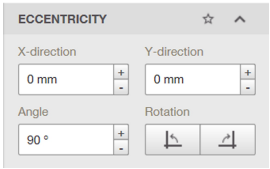
***** For internal use only ***** 13.12.2018 / 14:10:21 - H2\DOK-Projekt\System\PROFIS_Engineering\ENU\PROFIS_ING

8.2.5 Defining the steel profile

8.2.5.1 'Profile'

1. To define the rail profile, click on the **'Rail profiles'** button.
   ➥ The **'Select profile'** window is displayed.
2. To select the dimensions, click on the dimensions of the rail profile.
   ➥ The **'Select profile'** window is displayed.
3. To select the dimensions, click on the dimensions of the rail profile.
   ➥ The **'Select profile'** window is displayed.
4. To define the post profile, click on the **'Post profiles'** button.
   ➥ The **'Select profile'** window is displayed.
5. To select the dimensions, click on the dimensions of the post profile.
   ➥ The **'Select profile'** window is displayed.
6. Click on the appropriate profile size.

8.2.5.2 'Eccentricity'



1. To shift the steel profile, (X-/ and Y-axis), type the corresponding values in the **'X-direction'** and **'Y-direction'** input boxes.
2. To rotate the steel profile, enter the corresponding value in the **'Angle'** box.
3. To rotate the anchor plate to the left through 90°, click the button.
4. To rotate the anchor plate to the right through 90°, click the button.

8.2.6 Defining the statics

8.2.6.1 'Load cases'

- Activate the **'Show load inputs'** option button.

8.2.6.2 'Load type'

1. For static actions, select **'Static or quasi-static design'**.

   The load takes the action of static and predominantly stationary live loads into account.

2. Select one of the following option buttons:
   ➥ **'Design loads'**
   ➥ **'Characteristic loads'**

   'Design loads' – Enter the loads calculated at the design level.
   'Characteristic loads' – Enter the values for constant and variable loads. In PROFIS Engineering Suite these values are multiplied by the partial safety factors defined under **'Project options'**.

8.2.6.3 'Loading'

- To define the compressive stress on the masonry, enter the corresponding value in the **'Compressive stress on the wall'** input box.

***** For internal use only ***** 13.12.2018 / 14:10:21 - H2\DOK-Projekt\System\PROFIS_Engineering\ENU\PROFIS_ING

8.3 Fastening railings

8.3.1 Creating a favorites list

8.3.1.1 'My favorite inputs'

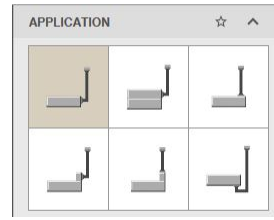
- To add a menu to the favorites inputs list, click this symbol ☆ for the desired menu.
  - The selected menu appears in the list of favorites.

To remove a menu from the favorite inputs list, click this option ★.

8.3.2 Defining the application

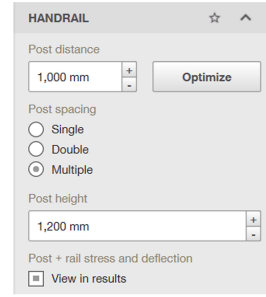
8.3.2.1 'Application'

- Select "Railing installation" from the **'Application'** section.
  - **'Concrete plate – Front'**
  - **'Concrete plate – Top'**
  - **'Concrete plate – Bottom'**
  - **'Stairs – Front'**
  - **'Balustrade – Outer'**
  - **'Balustrade – Top'**



8.3.2.2 'Handrail'

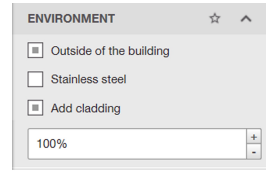
1. To define the distance between two railing posts, enter the value in the **'Post distance'** box.
2. To define the post system to be used, select the corresponding option button in the **'Post spacing'** section.
3. Enter the length of the post in the input box **'Post height'**.



4. To show the load, activate the **'Post + rail and deflection'** option button in the **'View in results'** section.

8.3.2.3 'Environment'

1. If the railing is to be installed on the exterior facade of the building, activate the **'Outside of the building'** option button.
2. If the railing is to be clad, activate the **'Add cladding'** option button and enter the cladding percentage in the input box.

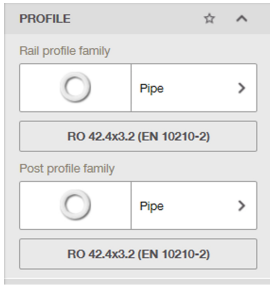


***** For internal use only ***** 13.12.2018 / 14:10:21 - H2\DOK-Projekt\System\PROFIS_Engineering\ENU\PROFIS_ING

8.3.3 Defining the steel profile

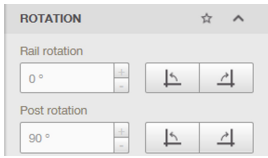
8.3.3.1 'Profile'

1. To define the rail profile, click on the **'Rail profiles'** button.
   ➥ The **'Select profile'** window is displayed.
2. To select the dimensions, click on the dimensions of the rail profile.
   ➥ The **'Select profile'** window is displayed.
3. To select the dimensions, click on the dimensions of the rail profile.
   ➥ The **'Select profile'** window is displayed.
4. To define the post profile, click on the **'Post profiles'** button.
   ➥ The **'Select profile'** window is displayed.
5. To select the dimensions, click on the dimensions of the post profile.
   ➥ The **'Select profile'** window is displayed.
6. Click on the appropriate profile size.



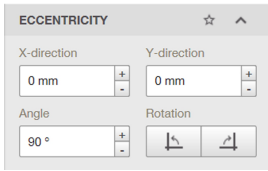
8.3.3.2 'Rotation'

1. To rotate the handrail, enter the appropriate value in the **'Handrail rotation'** input box.
2. To rotate the handrail 90° clockwise or counter-clockwise, click on the ⊿ button.
3. To rotate the post, enter the appropriate value in the **'Post rotation'** input box.
4. To rotate the post 90° clockwise, click on the ⊿ button.



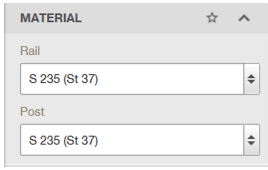
8.3.3.3 'Orientation'

- To shift the steel profile, (X and Y axes), enter the appropriate values in the **'X-direction'** and **'Y-direction'** input boxes.



8.3.3.4 'Material'

1. Select the corresponding railing from the **'Rail'** drop-down menu.
2. Select the corresponding material from the **'Post'** drop-down menu.



***** For internal use only **** 13.12.2018 / 14:10:21 - H2\DOK-Projekt\System\PROFIS_Engineering\ENU\PROFIS_ING

8.3.4 Defining the loads

8.3.4.1 'Load category'

1. Click the **'Load category'** button.
   ➥ The window **'Select load category'**
2. To define the type of building, click the applicable building.
3. To import more building types, click the **'Go to Hilti Online for more load category specifications'** button.

8.3.4.2 'Environment'

1. To define the wind load, click the **'Set windloads'** button.
   ➥ The **'Wind options'** window is displayed.
2. Select the wind zone from the **'Wind zone'** drop-down menu.
3. Select the terrain category from the **'Terrain category'** drop-down menu.
4. Enter the height of the railing in the **'Altitude above Ordinance Datum'** input box.
5. If a wind reduction is applicable, activate the **'Wind load reduction acc. to DIN EN 1991-1-4/NA, 7.2.10 '** option button.
6. Activate the corresponding option button in the **'Installation'** section.
7. Enter the corresponding values in the **'Height over ground (z)'**, **'Building width (w)'**, **'Building height (h)'** and **'Building length (L)'** input boxes.
8. Activate the corresponding option button in the **'Building zone'** section.
9. Click the **'Save'** button to confirm the entries.
10. Click the **'Cancel'** button to reject the entries.

8.3.4.3 'Dead load'

- Enter the self-weight in the **'Dead load'** input box.

8.3.4.4 'Loads'

- To display the load, activate the **'Show all loads'** option button.

8.3.4.5 'Load cases'

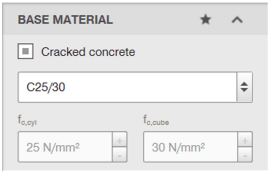
- Activate the **'Emergency exits & escape routes'** option button.

***** For internal use only ***** 13.12.2018 / 14:10:21 - H2\DOK-Projekt\System\PROFIS_Engineering\ENU\PROFIS_ING

8.3.5 Defining the base material

8.3.5.1 'Base material'

1. If the base material is cracked, activate the **'Cracked concrete'** option button.
2. Select the applicable concrete grade from the drop-down menu.



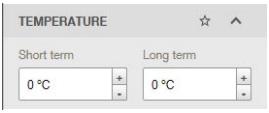
8.3.5.2 'Temperature'

1. In the **'Short term'** input box, enter the temperature to which the base material can be exposed for a short time.

   A short-term influence is understood to mean, for example, the temperature difference between day and night.

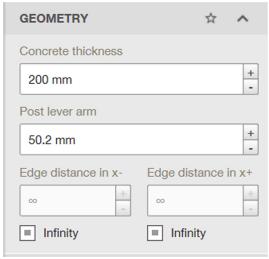
2. In the **'Long term'** input box, enter the long-term temperature influences.

   Long-term temperature influences are the base material's lasting temperature characteristics.



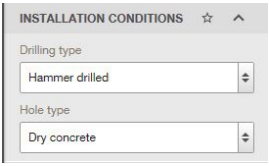
8.3.5.3 'Geometry'

1. Enter the thickness of the component in the **'Concrete thickness'** input box.
2. Enter the lever value for the post length in the **'Post lever arm'** input box.
3. To set the edge distance to "infinite", activate the **'Infinite'** option button.



8.3.5.4 'Installation conditions'

1. Select the drilling method from the **'Drilling method'** drop-down menu.
2. Select the condition of the drilled hole from the **'Hole type'** drop-down menu.



***** For internal use only **** 13.12.2018 / 14:10:21 - H2\DOK-Projekt\System\PROFIS_Engineering\ENU\PROFIS_ING

8.3.5.5 'Reinforcement'

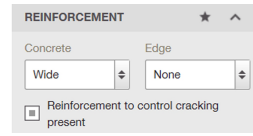
1. Select the spacing between reinforcing bars from the **'Concrete'** drop-down menu.

   The spacing is understood to be "wide" when the distance between bars is ≥150 mm, with any diameter, or when the distance is ≥100 mm with a diameter of ≤10 mm.

2. Select the spacing between rebars from the **'Edge'** drop-down menu.

   Straight edge reinforcement assumes edges reinforced with a rebar diameter of ≥12 mm. The hairpin reinforcement option assumes edge reinforcement with a bar diameter of ≥12 mm and closely spaced hairpin rebars at intervals of ≤100 mm. The edge reinforcement is taken into account in the verification of concrete edge breakage as a result of concrete cracking due to shear loading.

3. Activate the **'Reinforcement to control cracking present'** option button if the reinforced concrete is cracked. The maximum width of the cracks is approximately 3 mm.



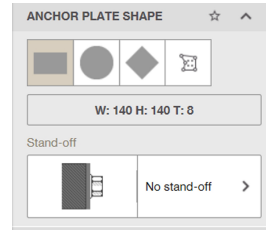
8.3.6 Defining the anchor plate

8.3.6.1 'Anchor plate position'

1. In the **'Rotation'** input box, enter the value by which you wish to rotate the anchor plate.
2. To rotate the anchor plate to the left through 90°, click the button.
3. To rotate the anchor plate to the right through 90°, click the button.

8.3.6.2 'Anchor plate shape'

1. Click the shape of the anchor you wish to use.
   ➥ The corresponding shape appears in the 3D editor.
2. To enter the dimensions of the anchor plate, click the button.
   ➥ The **'Anchor plate size'** window is displayed.
3. Type the required parameters in the corresponding input boxes.
4. Click **'Save'** to confirm the entries.
5. Click **'Cancel'** to cancel the operation.

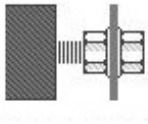
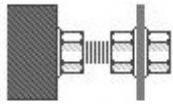
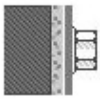


8.3.6.3 'Stand-off type'

- Select the type of stand-off installation.

	**'Not stand-off'**

***** For internal use only ***** 13.12.2018 / 14:10:21 - H2\DOK-Projekt\System\PROFIS_Engineering\ENU\PROFIS_ING

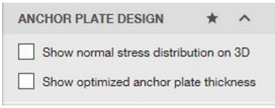
	**‘Stand-off without clamping’**
	**‘Stand-off with clamping’**
	**‘Stand-off with grounding’**

8.3.6.4 ‘Anchor plate thickness’

- Enter the thickness in the **‘Thickness’** box and press Enter to confirm your entry.

8.3.6.5 ‘Anchor plate design’

1. To show the load, activate the **‘Show normal stress distribution on 3D’** option button.
2. To optimize the anchor plate thickness, activate the **‘Show optimized anchor plate thickness’** option button.

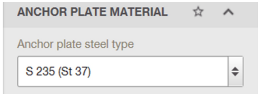


8.3.6.6 ‘Anchor plate offset’

- Enter the Offset Y eccentricity in the input box.

8.3.6.7 ‘Anchor plate material’

- Select the steel grade you are using from the drop-down box.



***** For internal use only **** 13.12.2018 / 14:10:21 - H2\DOK-Projekt\System\PROFIS_Engineering\ENU\PROFIS_ING

8.3.7 Defining the anchor

8.3.7.1 'Anchor'

1. To select the anchor type, click on the **'Family'** box.
   - ➥ The **'Select anchor'** window appears.

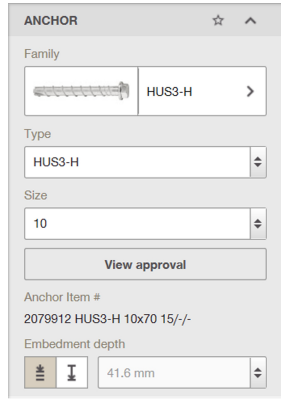
   Alternatively, you can click on the **'Family'** box in the main window to select the anchor type. If you want an anchor to be first in the **'Select anchor'** window you can highlight it as a favorite.

2. To search for an anchor, enter its name in the search box.
3. To sort the list of anchors, select the sorting criterion of your choice from the drop-down menu.
4. To calculate all anchors, click on the **'Calculate all'** button.
   - ➥ The degree of utilization and the geometry of the anchors are calculated and shown in the list.

   Click on **'Clear'** to return to the normal view.

5. To define the list of anchors, activate the appropriate radio button in the **'Filter'** box.
6. To filter the list of anchors by **'Fixture thickness'** or **'Hole diameter'**, enter the values in the **'max.'** and **'min.'** input boxes.
7. To filter the list of anchors by one of the following criteria, activate the appropriate radio button:
   - ➥ **'Anchor type'**
   - ➥ **'Corrosion / material'**
   - ➥ **'Cleaning'**
   - ➥ **'Setting'**
   - ➥ **'Thread type'**
   - ➥ **'Head configuration'**
   - ➥ **'Installation type'**
8. Select the anchorage depth in the **'Embedment depth'** box.
   - ➥ Optimized embedment depth –  .
   - ➥ User-selected embedment depth –  .

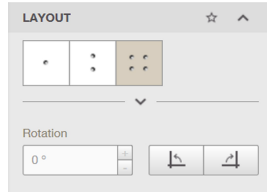
   Optimized anchorage depth – **Hilti** Anchor Installer determines the anchorage depth for maximum anchor loading capacity.
   User-defined anchorage depth – the user can specify the required anchorage depth. For adhesive mortar (chemical) anchors, all values are possible. For metal anchors, discrete values must be adhered to. You can select the appropriate depth here from the drop-down menu.



***** For internal use only ***** 13.12.2018 / 14:10:21 - H2\DOK-Projekt\System\PROFIS_Engineering\ENU\PROFIS_ING

8.3.7.2 'Layout'

1. Select the number and layout of the anchors in the **'Layout'** section.



> To apply a user-defined layout, click on the button.
> The 2D editor appears.

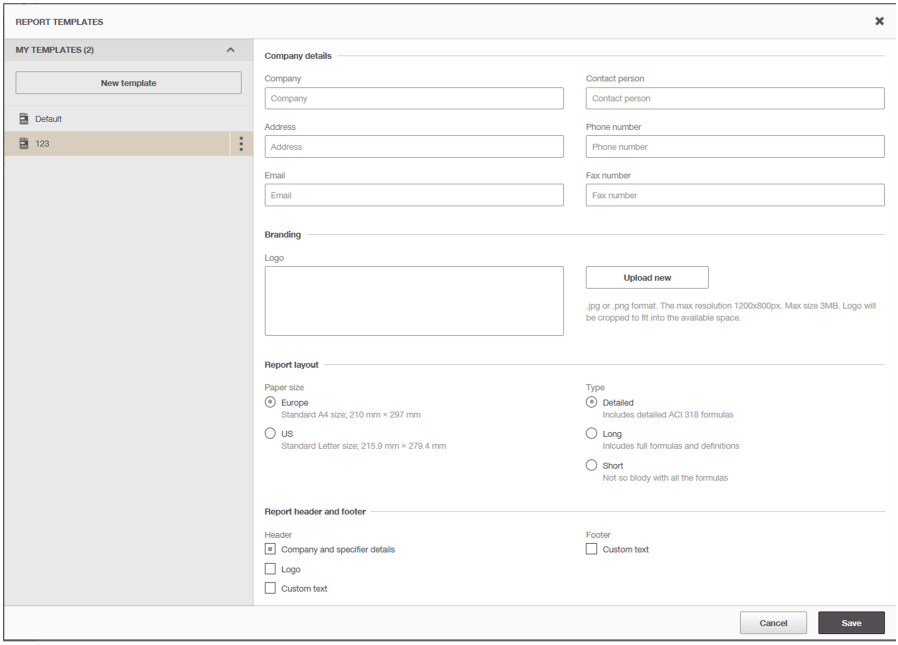
2. Enter the value by which you want to rotate the anchor plate In the **'Rotation'** input box.
3. To rotate the anchor plate 90° counter-clockwise, click on the button.
4. To rotate the anchor plate 90° clockwise, click on the button.

8.3.7.3 'Installation'

- To display the filled annular gap in the 3D editor, click the **'Filled holes'** button.

9 Report templates

9.1 Creating a new template



1. Click on the **'New template'** button in the **'My templates'** menu.
   ➥ An input box appears.
2. Enter a project name.
3. To confirm your entry, click on the button.
   ➥ The project appears in the list of templates.

***** For internal use only ***** 13.12.2018 / 14:10:21 - H2\DOK-Projekt\System\PROFIS_Engineering\ENU\PROFIS_ING

4. To reject the entry, click on the × button.
5. Enter the contact data in the input boxes in the **‘Company details’** section.
6. To upload a logo, click on the **‘Upload new’** button in the **‘Branding’** section.
   ➥ A window appears.
7. Navigate to the desired logo.
8. Confirm your selection.
   ➥ The selected logo appears in the **‘Logo’** section.
9. Select the desired language in the **‘Language’** drop-down menu.
10. Enter the desired value in the **‘First page number’** input box.
11. Activate the appropriate radio button in the **‘Page size’** section.
12. Activate the appropriate radio button in the **‘Type’** section.
13. Activate the appropriate checkboxes in the **‘Report header and footer’** section.
14. Enter your notes in the **‘Notes and comments’** section.
15. Click on the **‘Save’** button to confirm your entries.
16. Click the **‘Cancel’** button to reject the entries.

9.2 Renaming a template

1. Click the desired template in the **‘My templates’** menu.
   ➥ The ⋮ button is displayed.
2. Click the ⋮ button.
   ➥ The context menu is displayed.
3. Click **‘Rename’** in the context menu.
   ➥ The input box is displayed.
4. Enter the new project name.
5. Click the ✓ button to confirm the entries.
6. Click the × button to reject the entries.

9.3 Duplicating a template

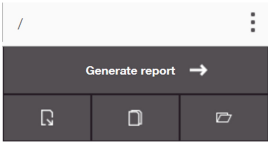
1. Click the desired template in the **‘My templates’** menu.
   ➥ The ⋮ button is displayed.
2. Click this button ⋮.
   ➥ The context menu is displayed.
3. Click **‘Duplicate’** in the context menu.
   ➥ The input box is displayed.
4. Enter the new project name.
5. Click the ✓ button to confirm the entries.
6. Click the × button to reject the entries.

9.4 Deleting a template

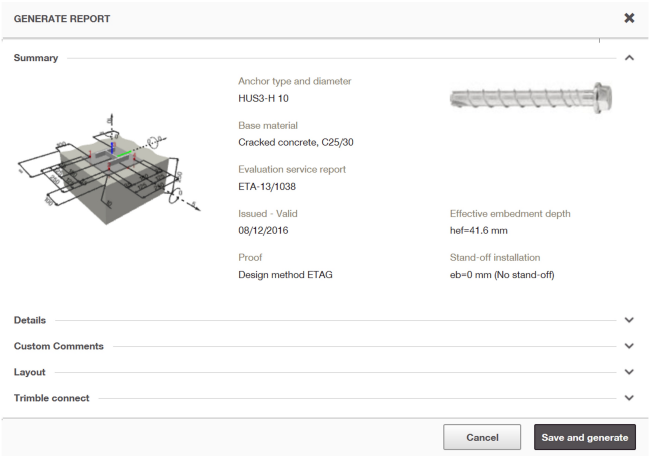
1. Click the desired template in the **‘My templates’** menu.
   ➥ The ⋮ button is displayed.
2. Click the ⋮ button.
   ➥ The context menu is displayed.
3. Click **‘Delete’** in the context menu.
   ➥ The template has been deleted.

***** For internal use only ***** 13.12.2018 / 14:10:21 - H2\DOK-Projekt\System\PROFIS_Engineering\ENU\PROFIS_ING

Changes made are saved automatically and do not need to be saved locally.



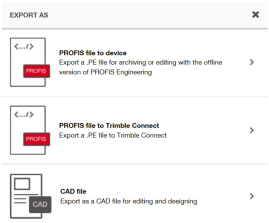
10.1 'Generate report'



1. To generate a report, click on the **'Generate report'** button.
   ➥ The **'Export'** window appears
2. To discard the report, click on **'Cancel'**.
3. To save and export the report, lick on the **'Save and export'** button.
   ➥ You can now save the report as a PDF file.

10.2 Exporting report

1. To export a report, click on the **'Export'** button.
   ➥ The **'Export as'** window appears
2. Select the desired format:
   ➥ PROFIS file
   ➥ PROFIS file to Trimble Connect
   ➥ CAD file



10.3 Duplicating dimensioning

- To duplicate dimensioning, click on the **'Duplicate dimensioning'** button.
  ➥ The project is opened in a new window.

10.4 Importing file

1. To import an existing file, click on the **'Import existing file'** button.
2. Select the file of your choice and click on **'Open'**.

***** For internal use only ***** 13.12.2018 / 14:10:21 - H2\DOK-Projekt\System\PROFIS_Engineering\ENU\PROFIS_ING

Hilti Corporation
LI-9494 Schaan
Tel.:+423 234 21 11
Fax:+423 234 29 65
www.hilti.group

Hilti = registered trademark of Hilti Corp., Schaan