Deltabeam preselection software – User Manual

The software runs ULS & SLS check for the selected Deltabeam type in ambient temperature i.e. no fire design is included. The user can only choose between different predetermined beam types, it is not possible to create new beam types. Torsion is not considered in the design.

The project consists of a single beam or a continuous beam line which consists of number of Deltabeams connected with the gerber-joint. The gerber-joint is constructed automatically between two successive Deltabeams. The continuous beam line is always horizontal and straight so that the floor is horizontal. Deltabeams can't be connected with the side connection.

Hollow-core unit is the only floor type in the software. However several types are available for a selection of countries. When choosing the Deltabeam type it advised to start with a type with equal depth as the hollow core units.
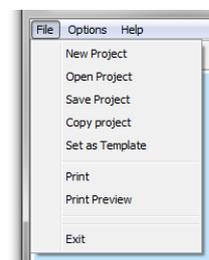
Information on Deltabeam gerber-joint and side connection, also other properties, is available in Peikko web page.

0 Menu

Icons from left to right: Open, Save, New project, Help and Analyze
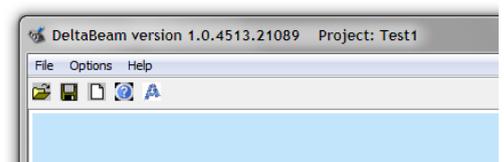
Menu item File includes eight functions which can be seen in figure on right.
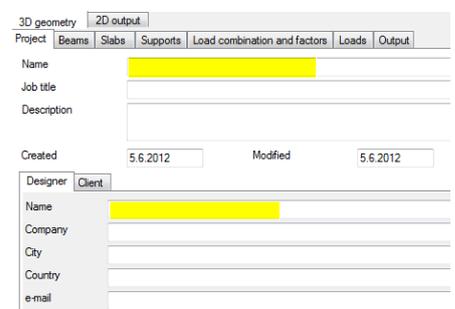

These functions are assumed to be familiar from other software's and are not explained further.

DESIGN PROCESS

The idea is to make the input for the project in the sequence of the tabs from left to right. In the following is explained shortly the main points of the input and also shown how the output looks.

1 Project data

To be able to save and print the project three 'Name' entries should be made. See the yellow line in the figure on right – the

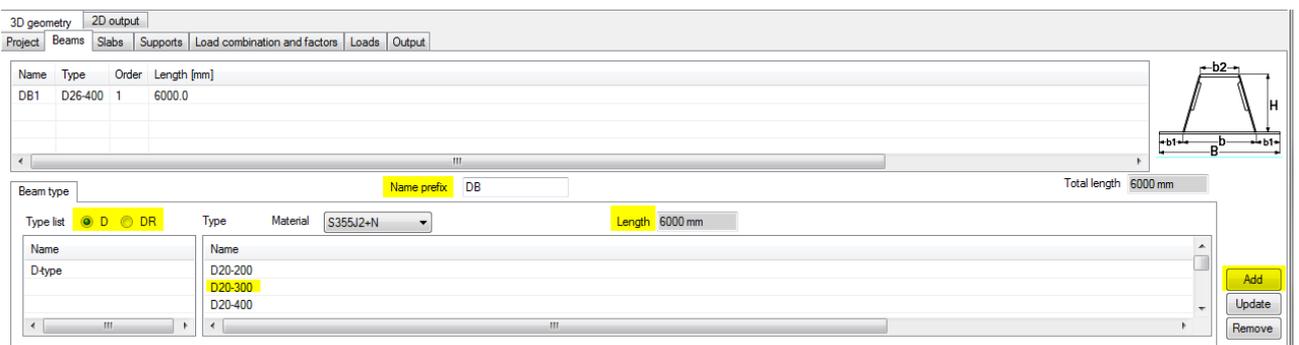third is in the Client tab, which is out of sight behind the Designer tab.

2 Beam data

To add a new beam select the 'Type list' item D or DR, Deltabeam type from the 'Name' list, add 'Name prefix', the length and push 'Add' button. These five points are indicated with yellow colour in the figure below. If more than one beam is added a gerber-joint is automatically assumed between the successive beams. Beams are always on a straight line in one plane.

To update information of an existing Deltabeam select the correct line in the beam list, revise the required information and push the 'Update' button.

To remove a beam from the project, select the correct line from the beam list and push 'Remove' button. The beam around the removed beam will be joined and supports that are not connected to any of the beams will be removed.

Adding a beam in between is not possible. Also changing the order of the beams in the model is not possible.

3 Slab data

To add a slab, select first the correct list from the 'Type list' and then the slab type. Input the prefix, choose the material, select the side and input the dimensions. Finally push 'Add' button. The inputs are indicated with yellow in the first figure below. If there is structural topping or U-profiles are needed they can be added in the second sub tab of the Slab tab; see the second figure below. Select the U-profile so that and slab top surface is flush with Deltabeam top flange.

To update information of an existing slab select the correct line in the slab list, revise the required information and push the 'Update' button.

To remove a slab from the project, select the correct line from the slab list and push 'Remove' button. The removal will not effect to the locations of the other slabs.
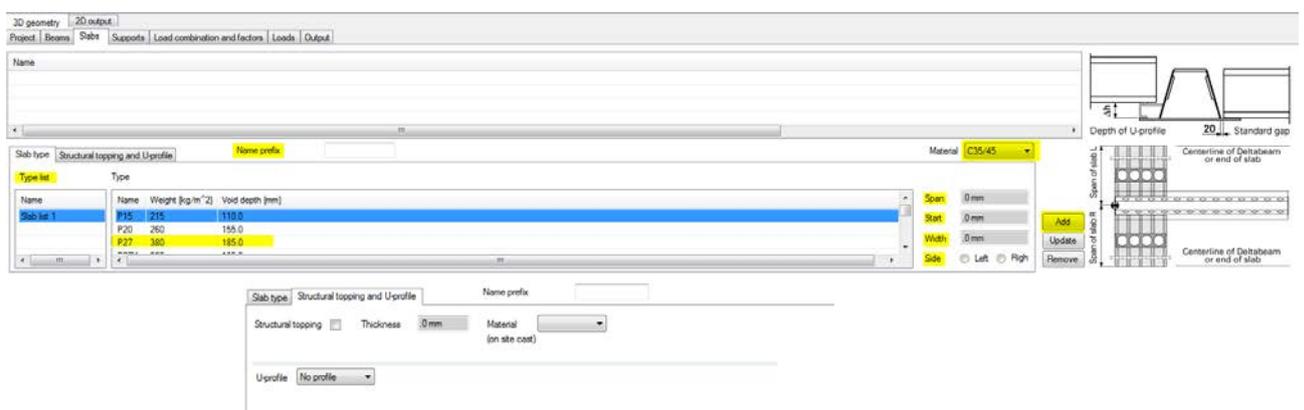
4 Supports

To add a new support input the distance from the origin and push 'Add' button. See the figure below.

To change the location of an existing support select the correct line in the support list, revise the distance and push the 'Update' button.

To remove a support from the project, select the correct line from the support list and push 'Remove' button. The removal will not effect to the locations of the other supports.

At all supports there is hinged connection to the Deltabeam i.e. it carries vertical reaction but no bending moments are transferred to the support.
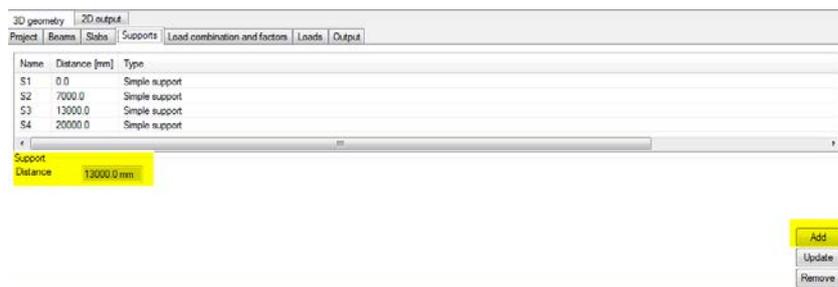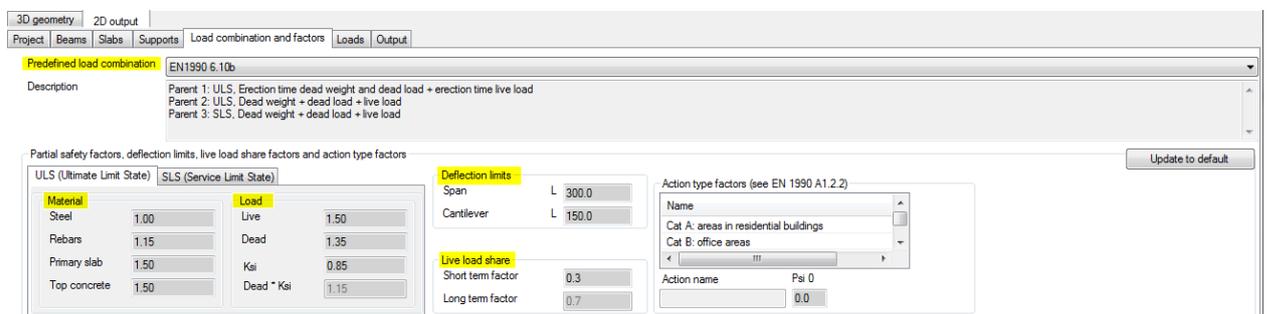

5 Safety factors and load combinations

Choose the load combination for your analysis from the list of predefined load combinations. Check that safety factors for materials and loads are correct. The inputs are indicated with yellow in the figure below.

The deflection limit is for live load. Live load share is to specify the short and long term portion of the live load; they are used to estimate the envelopes of internal forces.

For ksi (ξ) see table A1.2(B) NOTE 2 in EN 1990 annex A.

The dead load safety factor used in the software is in format: $\xi * \gamma_G$

so check value for ξ in your NA.

Example:

If the dead load safety factor your NA is 1.35, input

Dead: 1.35 (γ_G)

Ksi: 1.00 (ξ)   → Dead * Ksi = 1.35   (this value is used in the analysis)

There are two 'Predetermined load combinations' to be selected from. They both include three 'Parent combinations' for different design cases – each of them consists from 'Child combinations' which are the loads acting in the load history of the composite structure. The titles of the 'Predetermined load combinations' and the 'Parent combinations' are self explanatory – see the figure above. The user can choose at this point only one of the 'Predetermined load combinations' at the time – it is not possible to modify them or create new ones. After the analysis is done the user can view separately action effects of each 'Parent combination' and 'Child combination' in them.

6 Loads

To add a load tick first the box with title 'User defined', this activates the load input. Choose first 'Loading case', 'Action' and then 'Type'. Give unique name for the load then input the magnitude and the location finally push 'Add' button. The inputs are indicated with yellow in the figure below.

To revise an existing load select the correct line in the loads list, revise the required information and push the 'Update' button.

To remove a load from the project, select the correct line from the load list and push 'Remove' button. The removal will not effect to the locations or values of the other loads.

In the second sub tab 'Automatically generated characteristic loads' are the weights of the structure which are generated automatically by the software. They are used in the analysis; you should not input them again.
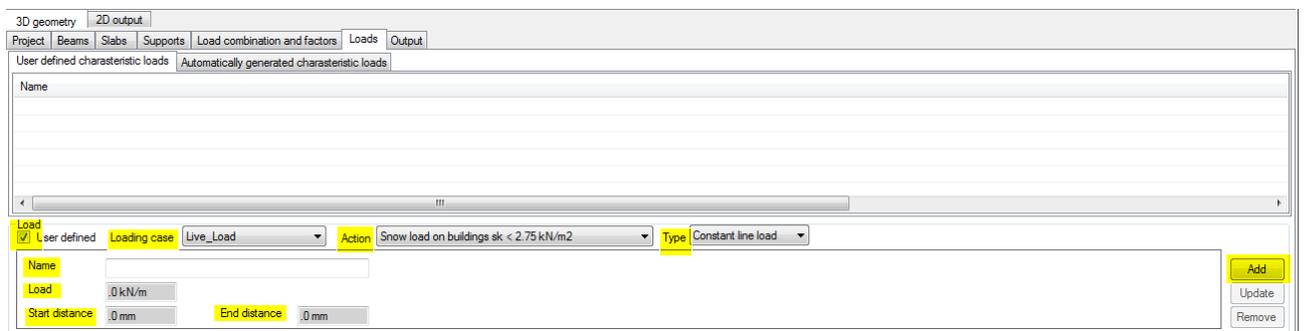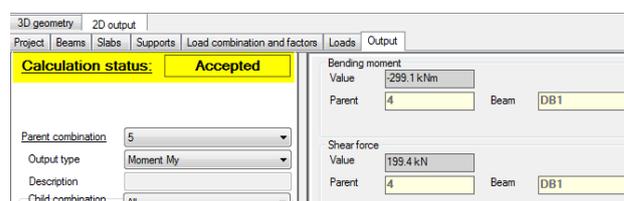

7 Output

After the analysis the results are viewed in graphical and numerical format. The result of the analysis is either ACCEPTED or REJECTED, see figure below.

REJECTED means that the beam type selected for the analysis is not adequate and it should be changed to a stronger one.

ACCEPTED means that the beam type selected for the analysis is strong enough but it does not tell if it is the most economical Deltabeam choice.

The action effects can be viewed by choosing the output type – different design cases can be viewed by choosing the 'Parent combination', see tab 'Load combination and factors' for explanations for the 'Parent combination'.

The results can be printed with the 'Print' or viewed with 'Print Preview' command.